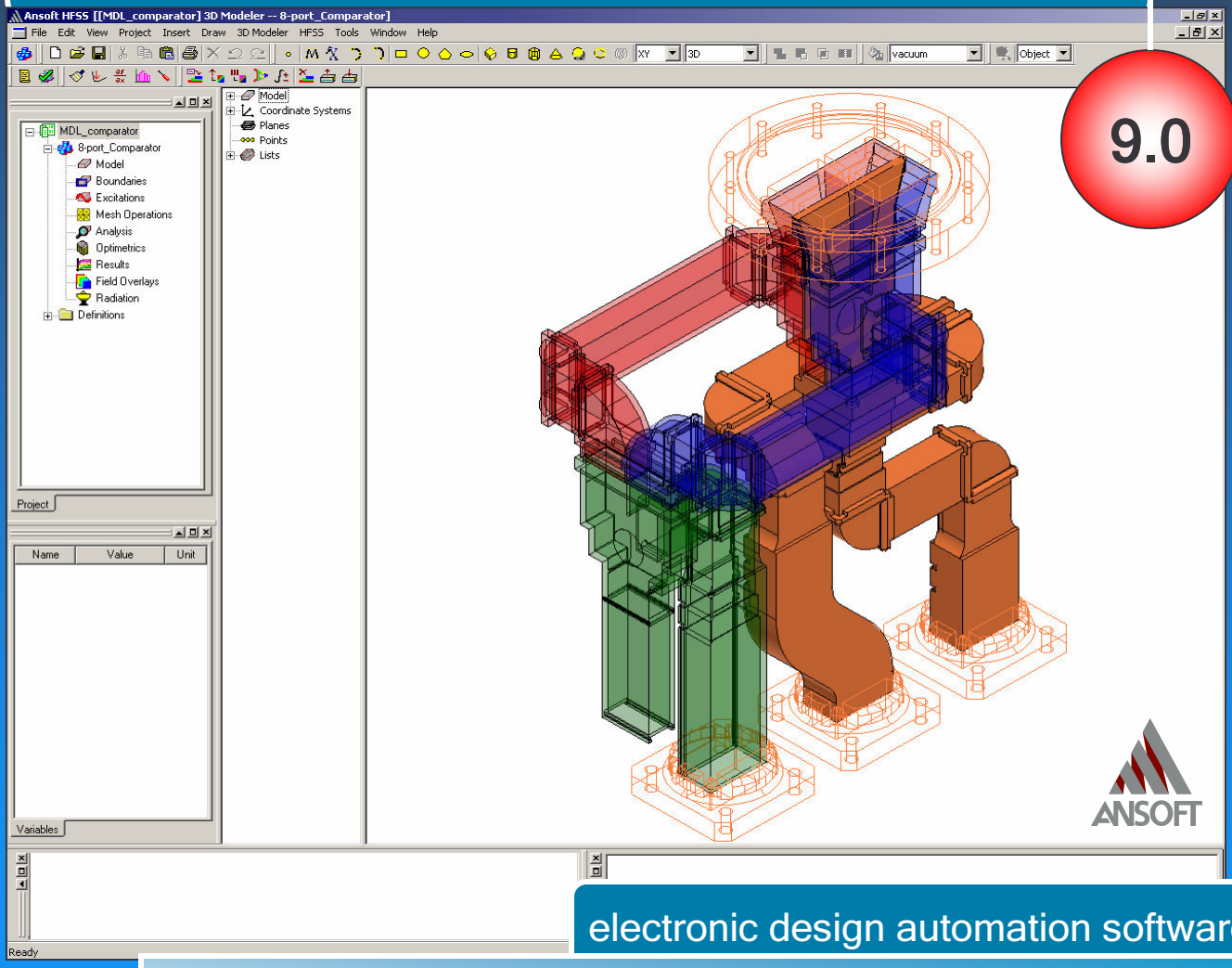
Ansoft

HFSS

High Frequency Structure Simulator



9.0

electronic design automation software

user's guide – High Frequency Structure Simulator

ANSOFT CORPORATION • Four Station Square Suite 200 • Pittsburgh, PA 15219-1119

This page is intentionally left blank.

The information contained in this document is subject to change without notice. Ansoft makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Ansoft shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

© 2003 Ansoft Corporation. All rights reserved.

Ansoft Corporation
Four Station Square
Suite 200
Pittsburgh, PA 15219
USA
Phone: 412-261-3200
Fax: 412-471-9427

HFSS and Optimetrics are registered trademarks or trademarks of Ansoft Corporation. All other trademarks are the property of their respective owners.

New editions of this manual will incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates which are incorporated at reprint do not cause the date to change.
Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Note that pages which are rearranged due to changes on a previous page are not considered to be revised.

Edition: REV1.0
Date: 08 August 2003
Software Version: 9.0

What is HFSS v 9.0?

- HFSS is a high-performance full-wave electromagnetic(EM) field simulator for arbitrary 3D volumetric passive device modeling that takes advantage of the familiar Microsoft Windows graphical user interface. It integrates simulation, visualization, solid modeling, and automation in an easy-to-learn environment where solutions to your 3D EM problems are quickly and accurately obtained. Ansoft HFSS employs the Finite Element Method(FEM), adaptive meshing, and brilliant graphics to give you unparalleled performance and insight to all of your 3D EM problems. Ansoft HFSS can be used to calculate parameters such as S-Parameters, Resonant Frequency, and Fields. Typical uses include:
  - Package Modeling – BGA, QFP, Flip-Chip
  - PCB Board Modeling – Power/Ground planes, Mesh Grid Grounds, Backplanes
  - Silicon/GaAs - Spiral Inductors, Transformers
  - EMC/EMI – Shield Enclosures, Coupling, Near- or Far-Field Radiation
  - Antennas/Mobile Communications – Patches, Dipoles, Horns, Conformal Cell Phone Antennas, Quadrafilar Helix, Specific Absorption Rate(SAR), Infinite Arrays, Radar Cross Section(RCS), Frequency Selective Surfaces(FSS)
  - Connectors – Coax, SFP/XFP, Backplane, Transitions
  - Waveguide – Filters, Resonators, Transitions, Couplers
  - Filters – Cavity Filters, Microstrip, Dielectric
- HFSS is an interactive simulation system whose basic mesh element is a tetrahedron. This allows you to solve any arbitrary 3D geometry, especially those with complex curves and shapes, in a fraction of the time it would take using other techniques.
- The name HFSS stands for High Frequency Structure Simulator. Ansoft pioneered the use of the Finite Element Method(FEM) for EM simulation by developing/implementing technologies such as tangential vector finite elements, adaptive meshing, and Adaptive Lanczos-Pade Sweep(ALPS). Today, HFSS continues to lead the industry with innovations such as Modes-to-Nodes and Full-Wave Spice™.
- Ansoft HFSS has evolved over a period of years with input from many users and industries. In industry, Ansoft HFSS is the tool of choice for high-productivity research, development, and virtual prototyping.

Contents

- This document discusses some basic concepts and terminology used throughout the Ansoft HFSS application. It provides an overview of the following topics:
  0. Fundamentals
     - Ansoft HFSS Desktop
     - Opening a Design
     - Setting Model Type
  1. Parametric Model Creation
     - 1.1 Boundary Conditions
     - 1.2 Excitations
  2. Analysis Setup
  3. Data Reporting
  4. Solve Loop
     - 4.1 Mesh Operations
  5. Examples - Antenna
  6. Examples - Microwave
  7. Examples - Signal Integrity
  8. Examples - EMC/EMI

Installing the Ansoft HFSS v 9.0 Software

System Requirements

- Microsoft Windows XP, Windows 2000, or Windows NT4.0. For up-to-date information, refer to the HFSS v.9.0 Release Notes.
- Pentium -based computer
- 128MB RAM minimum
- 8MB Video Card minimum
- Mouse or other pointing device
- CD-ROM drive

> **NOTE**: You should make backup copies of all HFSS projects created with a previous version of the software before opening them in HFSS v.9.0

Installing the Ansoft HFSS v.9.0 Software

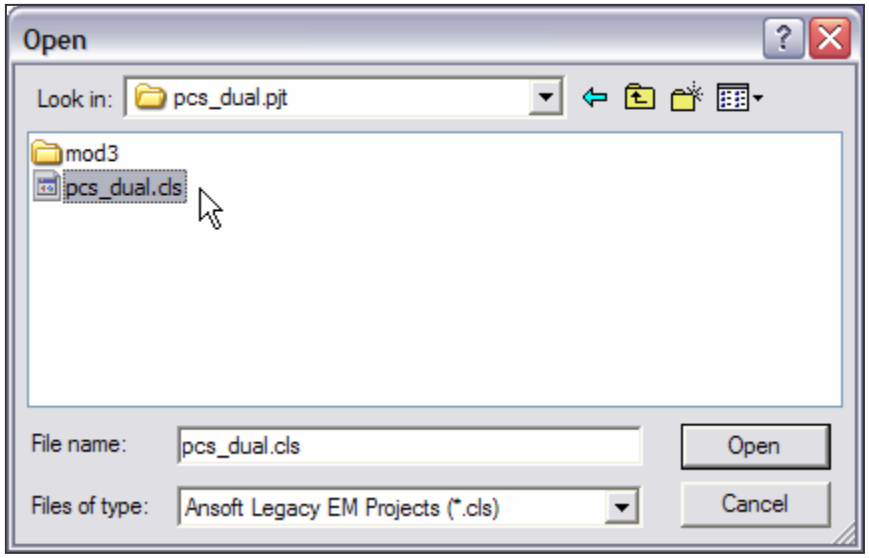
- For up-to-date information, refer to the HFSS v.9.0 Installation Guide

Starting Ansoft HFSS

1. Click the Microsoft **Start** button, select **Programs**, and select the **Ansoft, HFSS 9** program group. Click **HFSS 9**.
2. **Or** Double click on the HFSS 9 icon on the Windows Desktop

Converting Older HFSS file to HFSS v.9.0

- Because of changes to the HFSS files with the development of HFSS v.9.0, opening a HFSS document from an earlier release may take more time than you are used to experiencing. However, once the file has been opened and saved, subsequent opening time will return to normal
- Ansoft HFSS v.9.0 provides a way for you to automatically convert your HFSS projects from an earlier version to the HFSS v.9.0 format.
- To access HFSS projects in an earlier version.
  - **From HFSS v9.0**,
    1. Select the menu item ***File > Open***
    2. Open dialog
       1. Files of Type: **Ansoft Legacy EM Projects (.cls)**
       2. Browse to the existing project and select the .cls file
       3. Click the **Open** button



Getting Help

- If you have any questions while you are using Ansoft HFSS v.9.0 you can find answers in several ways:
  - **Ansoft HFSS Online Help** provides assistance while you are working.
    - To get help about a specific, active dialog box, click the **Help** button in the dialog box or press the **F1** key.
    - Select the menu item ***Help > Contents*** to access the online help system.
    - **Tooltip**s are available to provide information about tools on the toolbars or dialog boxes. When you hold the pointer over a tool for a brief time, a tooltip appears to display the name of the tool.
    - As you move the pointer over a tool or click a menu item, the **Status Bar** at the bottom of the Ansoft HFSS window provides a brief description of the function of the tool or menu item.
    - The Ansoft HFSS v.9.0 Getting Started guide provides detailed information about using HFSS to create and solve 3D EM projects.
  - **Ansoft Technical Support**
    - To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft corporate website, www.ansoft.com and select **Contact**.
      - Your Ansoft sales engineer may also be contacted in order to obtain this information.

Visiting the Ansoft Web Site

- If your computer is connected to the Internet, you can visit the Ansoft Web site to learn more about the Ansoft company and products.
  - From the Ansoft Desktop
    - Select the menu item ***Help > Ansoft Corporate Website*** to access the Online Technical Support (OTS) system.
  - From your Internet browser
    - Visit www.ansoft.com

For Technical Support

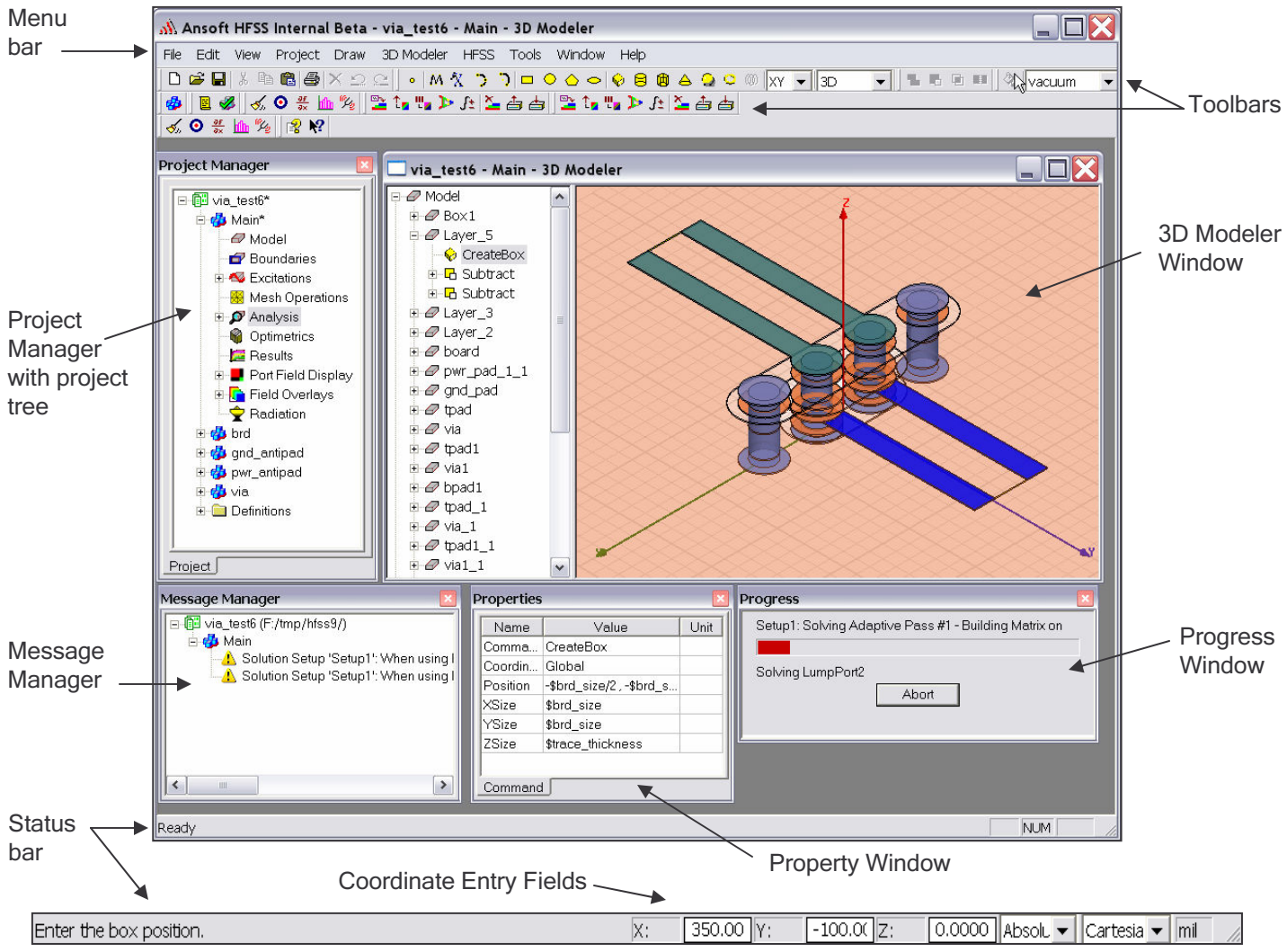
- The following link will direct you to the Ansoft Support Page. The Ansoft Support Pages provide additional documentation, training, and application notes.
  - Web Site: http://www.ansoft.com/support.cfm
  - Technical Support:
    - 9-4 EST: (412) 261-3200 x0 – Ask for Technical Support

Application Engineers for North America

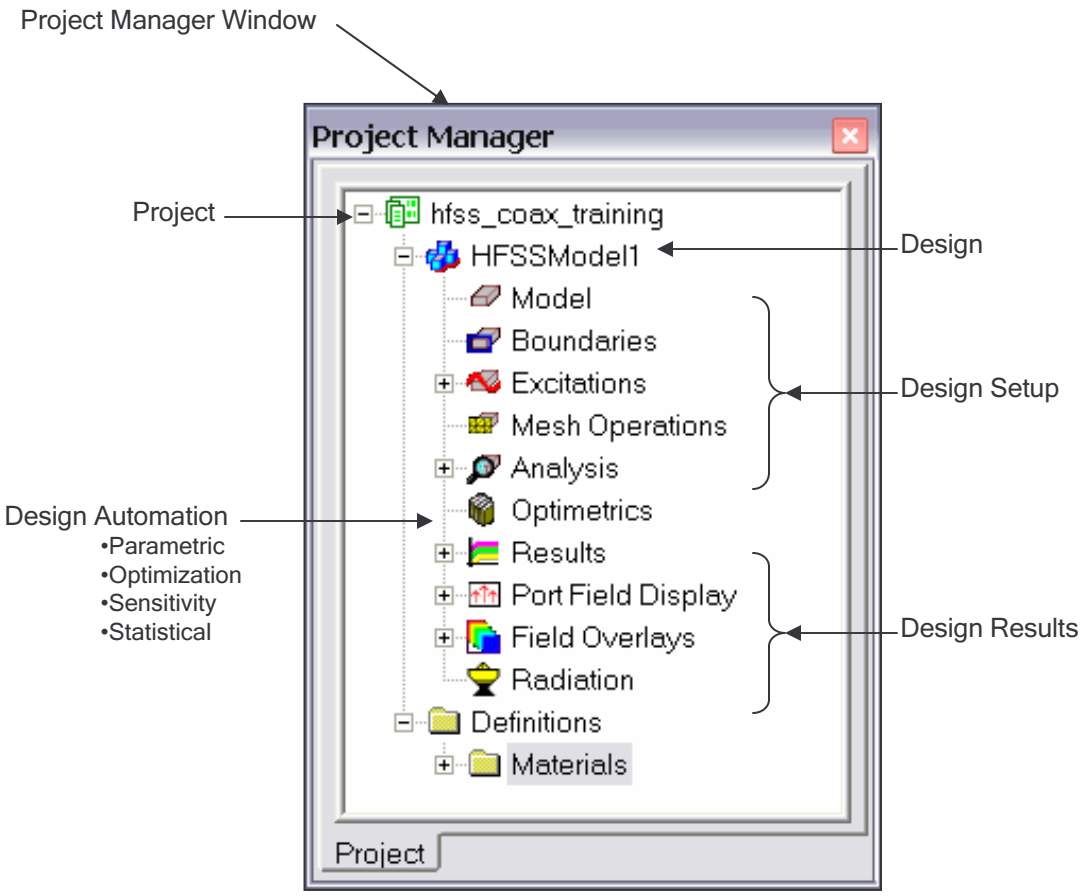
- The names and numbers in this list may change without notice
  - **Ansoft Office: Pittsburgh, PA, 412-261-3200**
    - Yianni Antoniadis, x175, yantoniadis@ansoft.com
    - Shu Li, x172, sli@ansoft.com
  - **Ansoft Office: Boston, MA, 781-229-8900**
    - Jim Delap, x133, jdelap@ansoft.com
  - **Ansoft Office: Boulder, CO, 303-541-9525**
    - Bryan Boots, x31, bboots@ansoft.com
    - Brian Gray, x32, bgray@ansoft.com
  - **Ansoft Office: El Segundo, CA, 310-426-2287**
    - Aaron Edwards, aedwards@ansoft.com
    - Liza Ma, lma@ansoft.com
    - Peter Shin, pshin@ansoft.com
  - **Ansoft Office: Santa Clara, CA, 408-261-9095**
    - Harpreet Randhawa, x210, hrandhawa@ansoft.com
    - Denis Soldo, x214, dsoldo@ansoft.com
  - **Others**
    - Matt Commens, IL, 847-925-9066, mcommens@ansoft.com
    - Lisa Murphy, AZ, 408-288-1114, lmurphy@ansoft.com
    - Jim Sherman, FL, 727-394-1023, jsherman@ansoft.com
  - **Canada**
    - Markus Kopp, Ottawa, 613-241-0500, mkopp@ansoft.com

Ansoft Terms

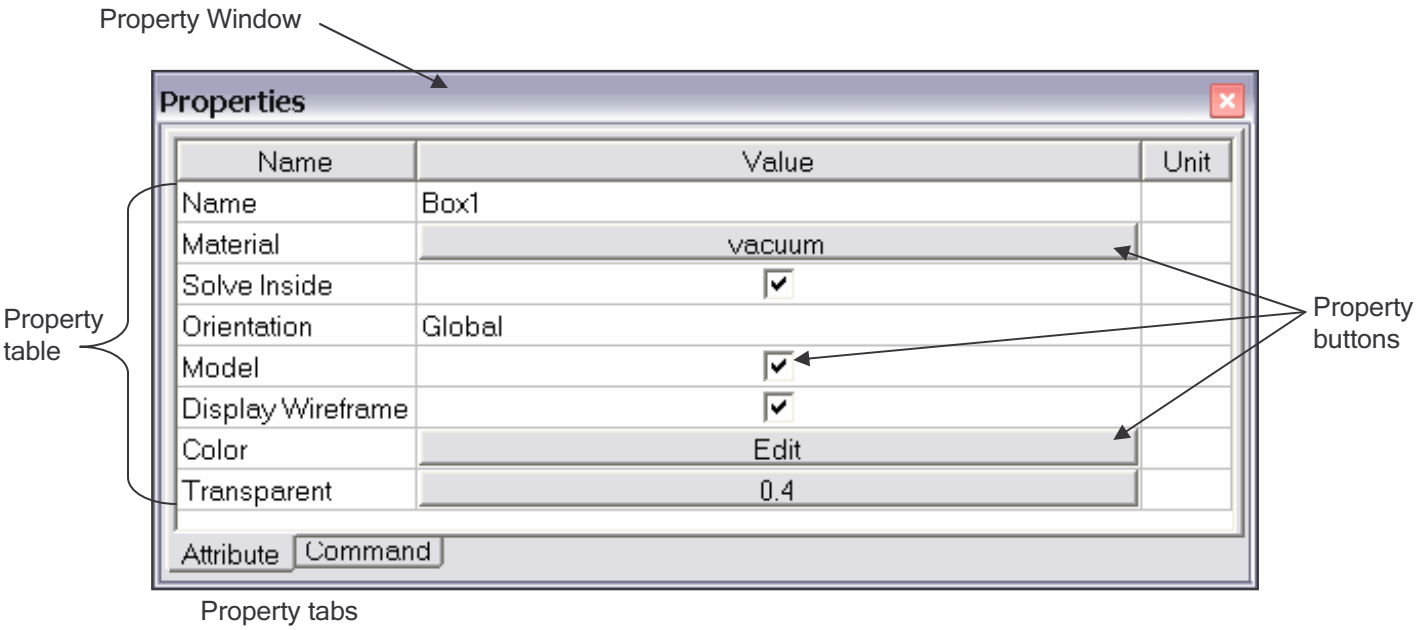
- The Ansoft HFSS window has several optional panels:
  - A **Project Manager** which contains a design tree which lists the structure of the project.
  - A **Message Manager** that allows you to view any errors or warnings that occur before you begin a simulation.
  - A **Property Window** that displays and allows you to change model parameters or attributes.
  - A **Progress Window** that displays solution progress.
  - A **3D Modeler Window** which contains the model and model tree for the active design. For more information about the3D Modeler Window, see chapter 1.



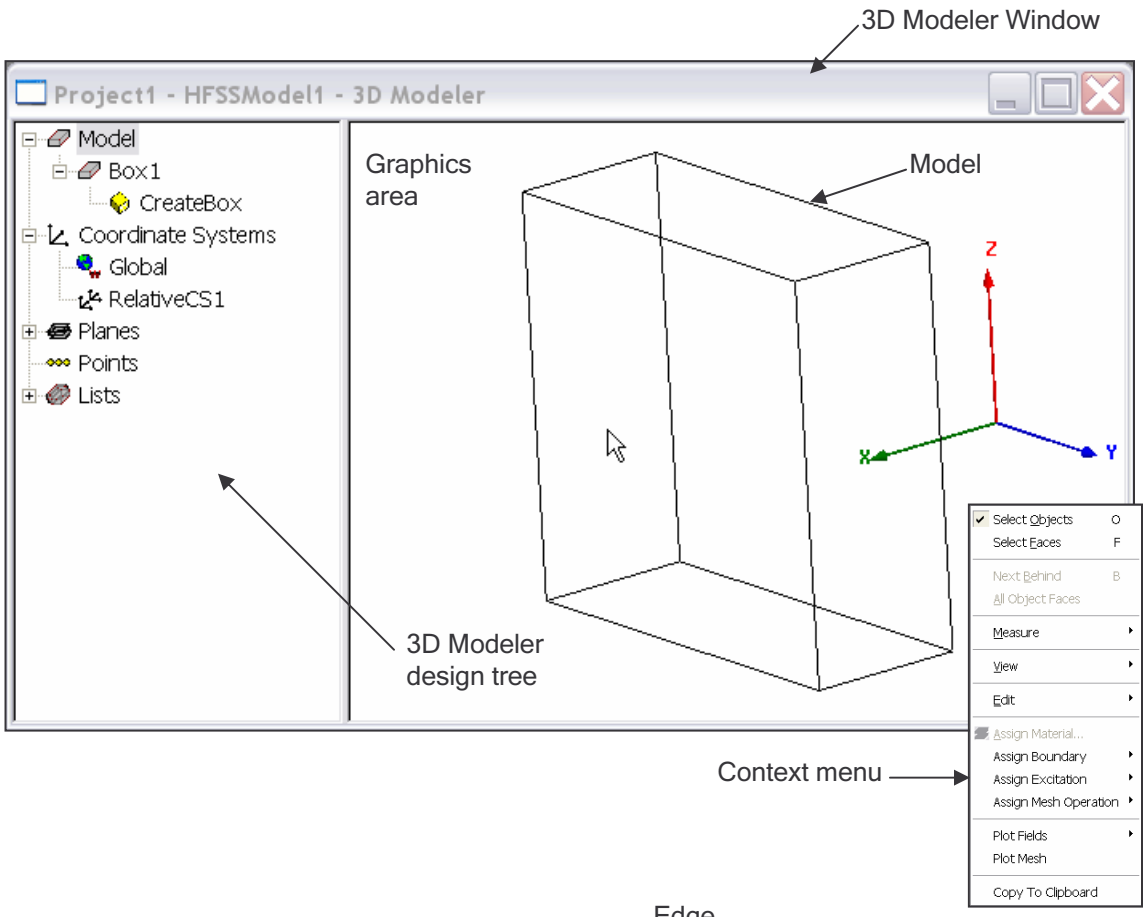
Project Manager

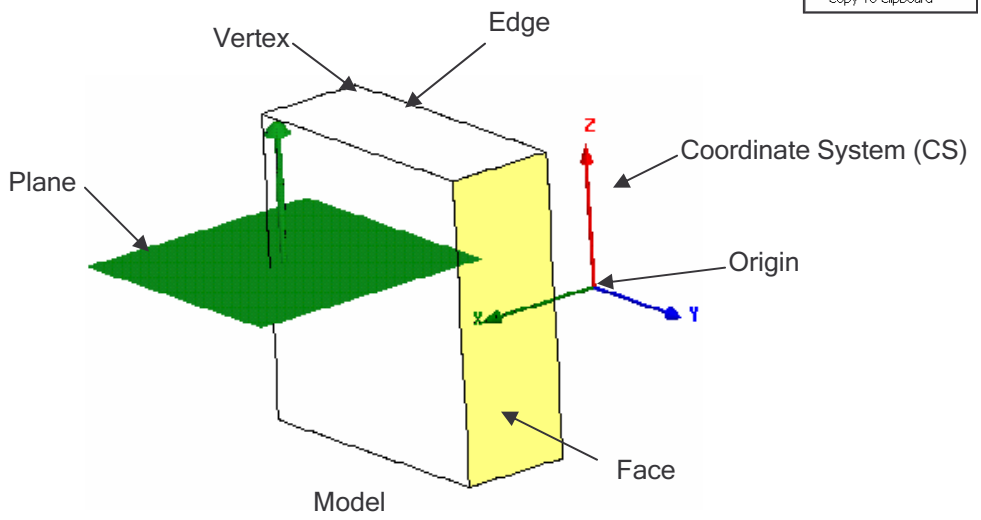


Property Window

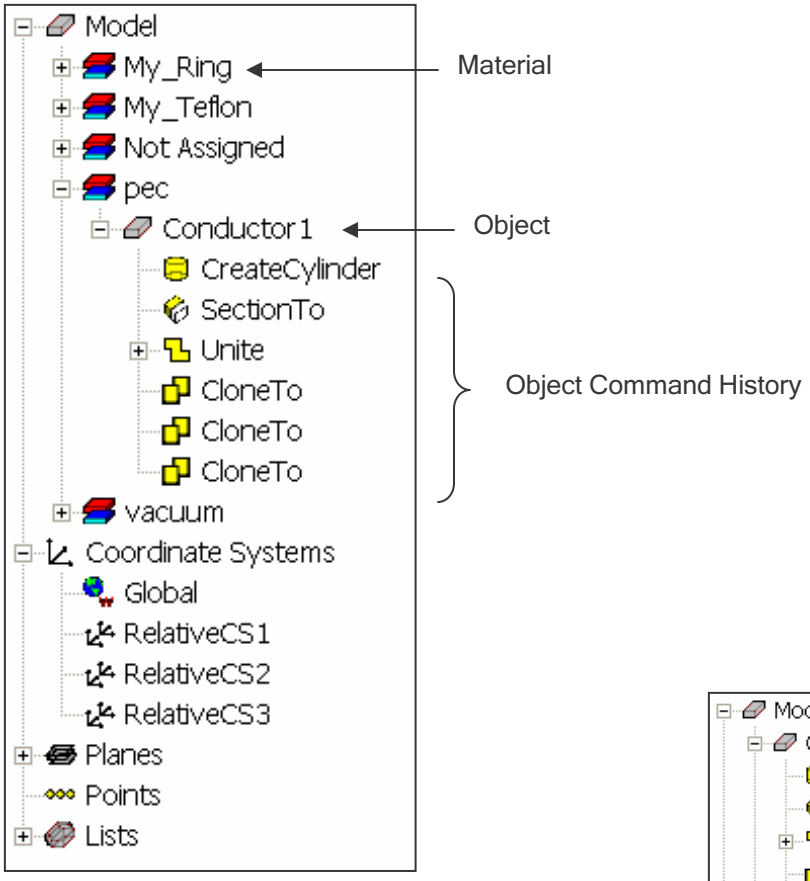


Ansoft 3D Modeler

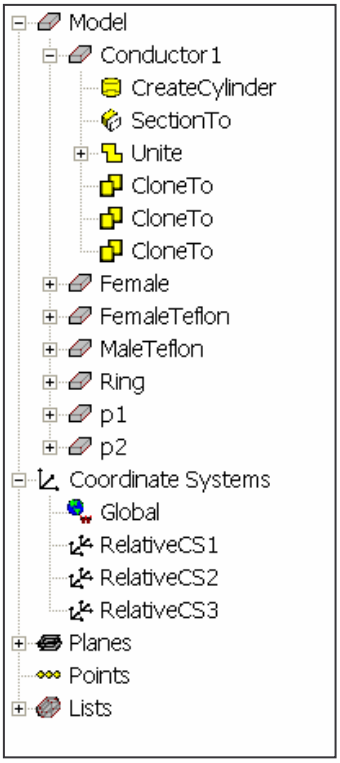




3D Modeler Design Tree



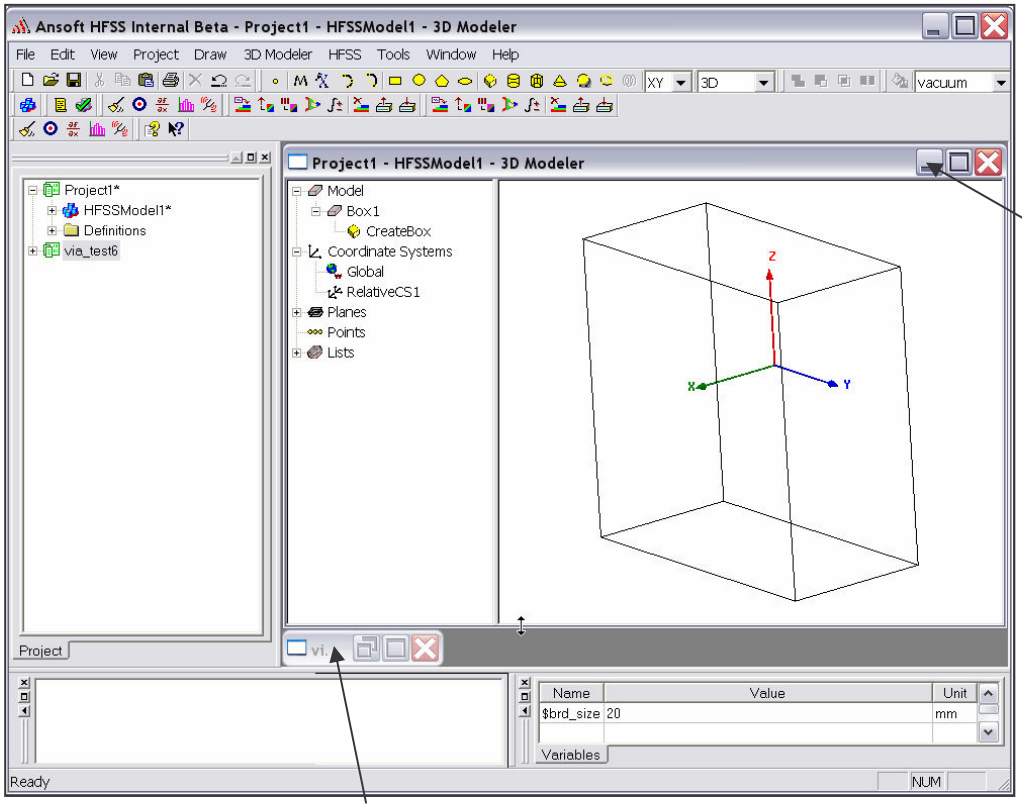
Grouped by Material



Object View

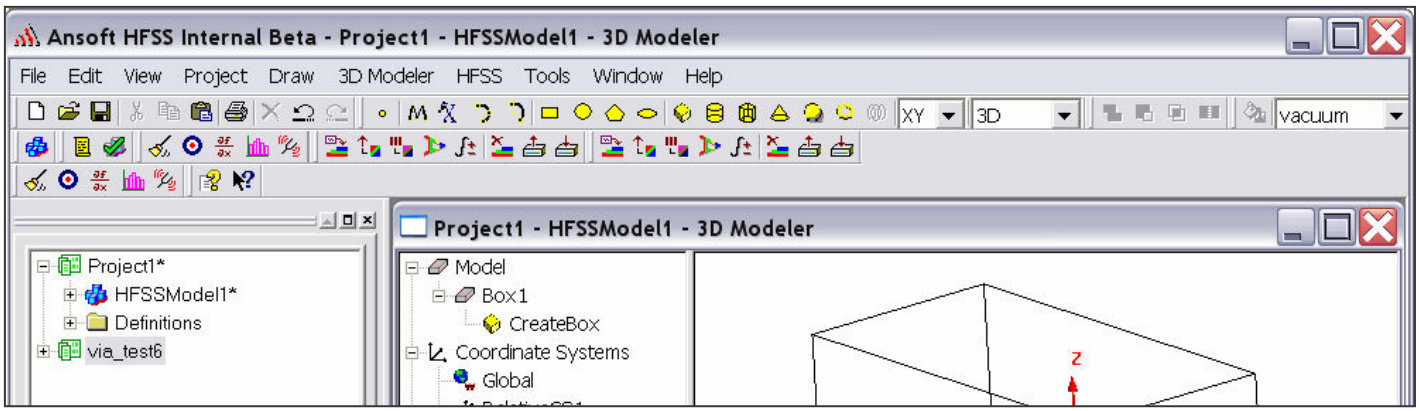
Design Windows

- In the Ansoft HFSS Desktop, each project can have multiple designs and each design is displayed in a separate window.
- You can have multiple projects and design windows open at the same time. Also, you can have multiple views of the same design visible at the same time.
- To arrange the windows, you can drag them by the title bar, and resize them by dragging a corner or border. Also, you can select one of the following menu options: ***Window >Cascade***, ***Window >Tile Vertically***, or ***Window > Tile Horizontally.***
- To organize your Ansoft HFSS window, you can iconize open designs. Click the Iconize ** symbol in the upper right corner of the document border. An icon appears in the lower part of the Ansoft HFSS window. If the icon is not visible, it may be behind another open document. Resize any open documents as necessary. Select the menu item ***Window > Arrange Icons*** to arrange them at the bottom of the Ansoft HFSS window.
- Select the menu item ***Window > Close All*** to close all open design. You are prompted to **Save** unsaved designs.



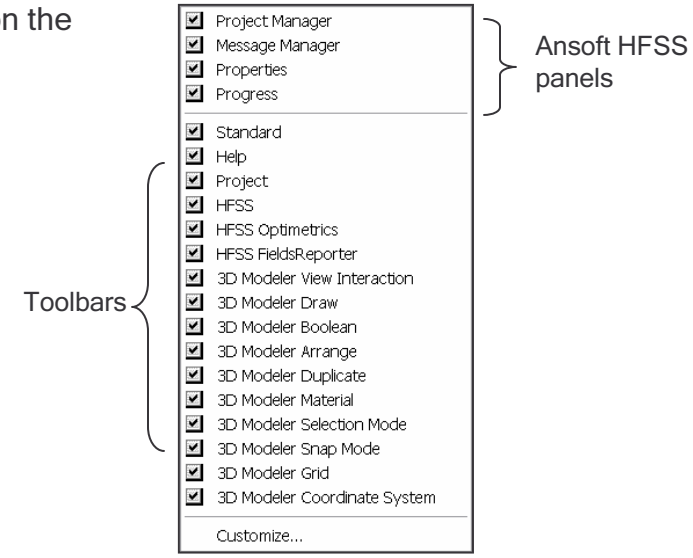
Toolbars

- The toolbar buttons are shortcuts for frequently used commands. Most of the available toolbars are displayed in this illustration of the Ansoft HFSS initial screen, but your Ansoft HFSS window probably will not be arranged this way. You can customize your toolbar display in a way that is convenient for you.
- Some toolbars are always displayed; other toolbars display automatically when you select a document of the related type. For example, when you select a 2D report from the project tree, the 2D report toolbar displays.



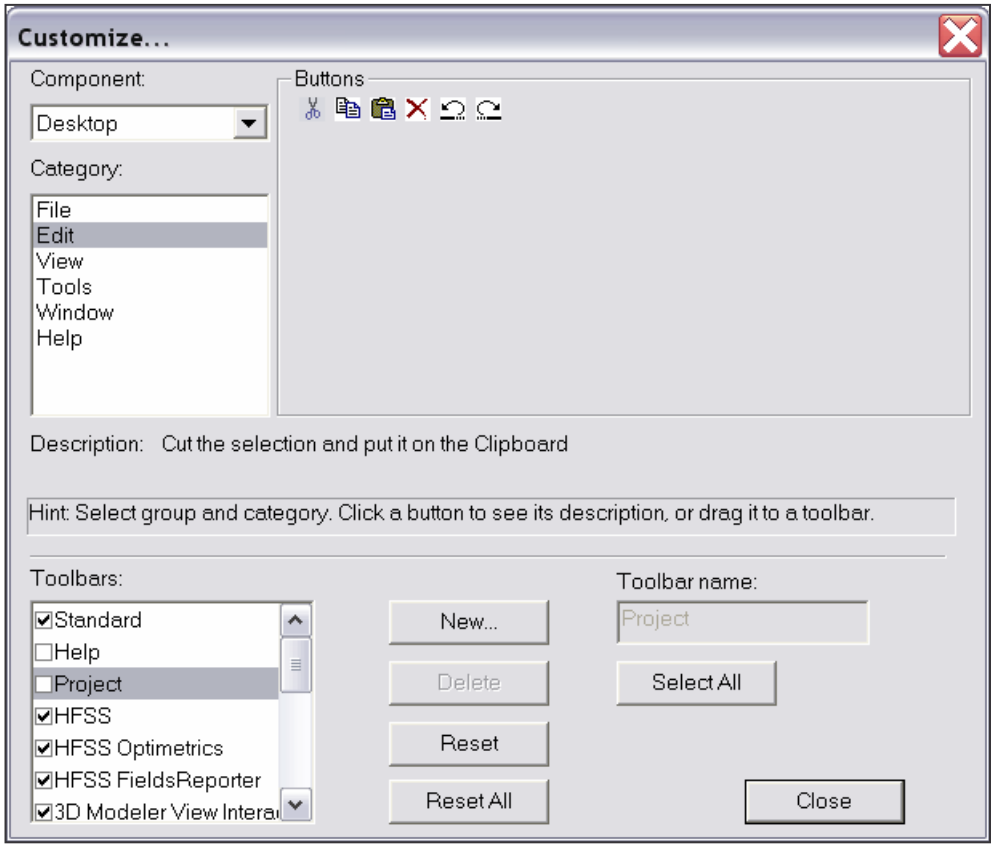
To display or hide individual toolbars:

- Right-click the Ansoft HFSS window frame.
  - A list of all the toolbars is displayed. The toolbars with a check mark beside them are visible; the toolbars without a check mark are hidden. Click the toolbar name to turn its display on or off
- To make changes to the toolbars, select the menu item ***Tools > Customize***. See **Customize and Arrange Toolbars** on the next page.



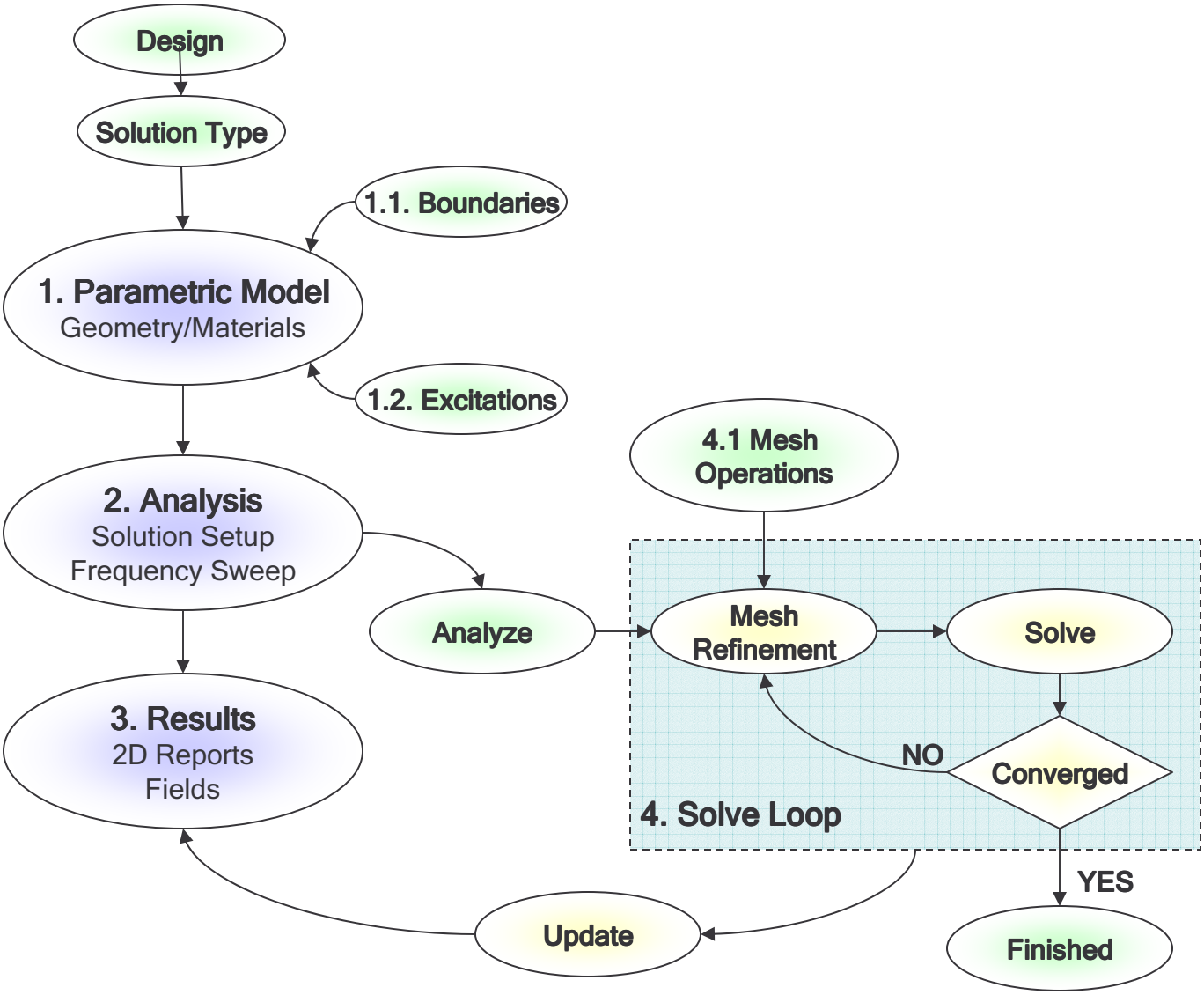
Customize and Arrange Toolbars

- To customize toolbars:
  - Select the menu item ***Tools > Customize,*** or right-click the Ansoft HFSS window frame and click **Customize** at the bottom of the toolbar list.
  - In the Customize dialog, you can do the following:
    - **View a Description of the toolbar commands**
      1. Select an item from the Component pull-down list
      2. Select an item from the Category list
      3. Using the mouse click on the Buttons to display the Description
      4. Click the **Close** button when you are finished
    - **Toggle the visibility of toolbars**
      1. From the Toolbar list, toggle the check boxes to control the visibility of the toolbars
      2. Click the **Close** button when you are finished



Ansoft HFSS Desktop

- The Ansoft HFSS Desktop provides an intuitive, easy-to-use interface for developing passive RF device models. Creating designs, involves the following:
  1. **Parametric Model Generation** – creating the geometry, boundaries and excitations
  2. **Analysis Setup** – defining solution setup and frequency sweeps
  3. **Results** – creating 2D reports and field plots
  4. **Solve Loop** - the solution process is fully automated
- To understand how these processes co-exist, examine the illustration shown below.



Opening a HFSS project

This section describes how to open a new or existing project.

Opening a New project

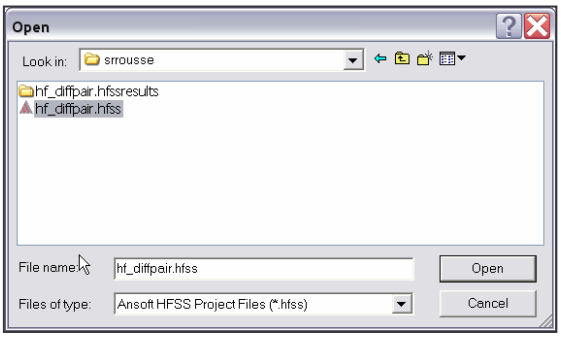
To open a new project:

1. In an Ansoft HFSS window, select the menu item ***File > New***.
2. Select the menu ***Project > Insert HFSS Design***.

Opening an Existing HFSS project

To open an existing project:

1. In an Ansoft HFSS window, select the menu ***File > Open***. Use the Open dialog to select the project.
2. Click **Open** to open the project



Opening an Existing Project from Explorer

- You can open a project directly from the Microsoft Windows Explorer.
- **To open a project from Windows Explorer, do one of the following:**
  - Double-click on the name of the project in Windows Explorer.
  - Right-click the name of the project in Windows Explorer and select **Open** from the shortcut menu.

Set Solution Type

This section describes how to set the Solution Type. The Solution Type defines the type of results, how the excitations are defined, and the convergence. The following Solution Types are available:

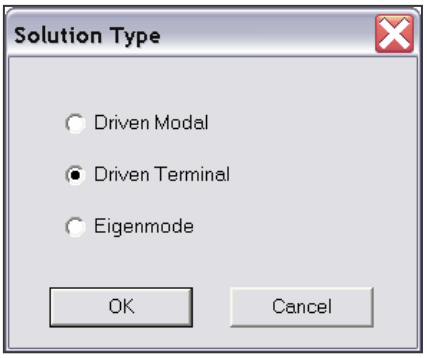
1. **Driven Modal** - calculates the modal-based S-parameters. The S-matrix solutions will be expressed in terms of the incident and reflected powers of waveguide modes.
2. **Driven Terminal** - calculates the terminal-based S-parameters of multi-conductor transmission line ports. The S-matrix solutions will be expressed in terms of terminal voltages and currents.
3. **Eignemode** – calculate the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies.

Convergence

- **Driven Modal** – Delta S for modal S-Parameters. This was the only convergence method available for Driven Solutions in previous versions.
- **Driven Terminal***New* – Delta S for the single-ended or differential nodal S-Parameters.
- **Eigenmode** - Delta F

To set the solution type:

1. Select the menu item ***HFSS > Solution Type***
2. Solution Type Window:
    1. Choose one of the following:
        1. **Driven Modal**
        2. **Driven Terminal**
        3. **Eigenmode**
    2. Click the **OK** button

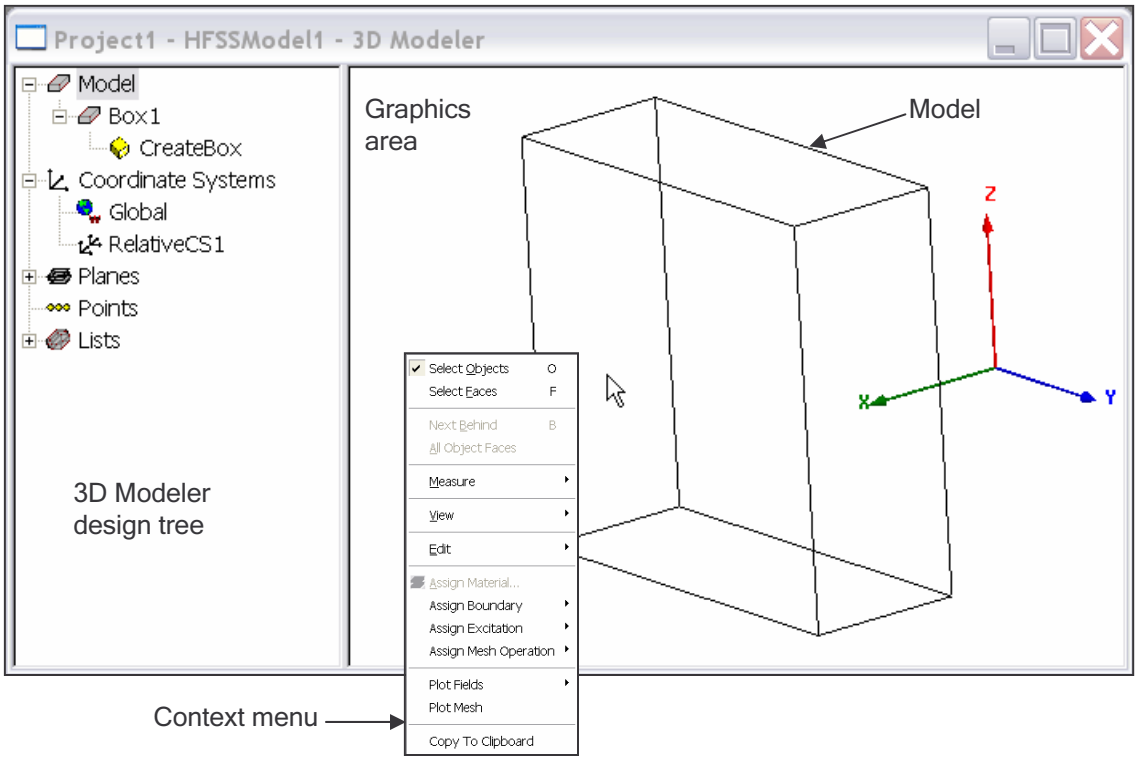


Parametric Model Creation

- The Ansoft HFSS 3D Modeler is designed for ease of use and flexibility. The power of the 3D Modeler is in its unique ability to create fully parametric designs without editing complex macros/model history.
- The purpose of this chapter is to provide an overview of the 3D Modeling capabilities. By understanding the basic concepts outlined here you will be able to quickly take advantage of the full feature set offered by the 3D Parametric Modeler.

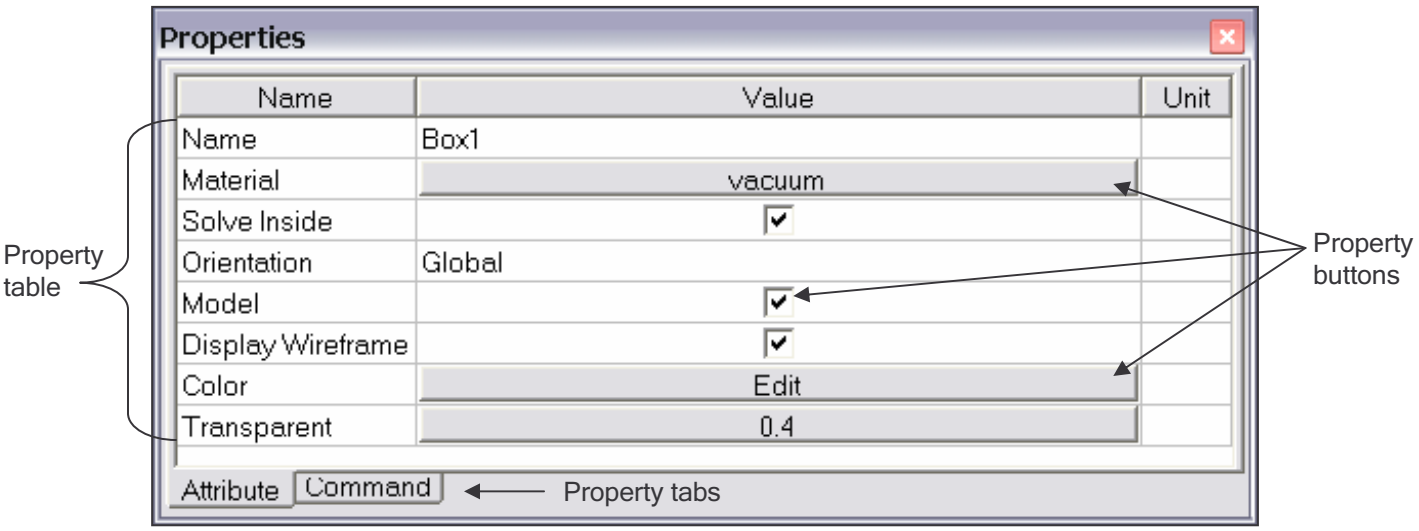
Overview of the 3D Modeler User Interface

- The following picture shows the 3D Modeler window.
  - **3D Modeler Design Tree** – The 3D Modeler Design Tree is an essential part of the user interface. From here you may access the structural elements in addition to any object dependencies and attributes.
  - **Context Menus** – Context menus are a flexible way of accessing frequently used menu commands for the current context. The contents of these menus change dynamically and are available throughout the interface by clicking the right mouse button.
  - **Graphics Area** – The graphics area is used to interact with the structural elements.

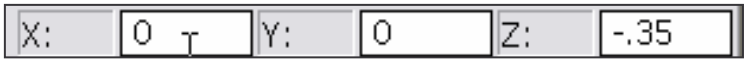


Overview of the 3D Modeler User Interface (Continued)

- When using the 3D Modeler interface you will also interact with two additional interfaces:
  - **Property Window** – The Property Window is used to view or modify the attributes and dimensions of structural objects



  - **Status Bar/Coordinate Entry** – The Status Bar on the Ansoft HFSS Desktop Window displays the Coordinate Entry fields that can be used to define points or offsets during the creation of structural objects



Grid Plane

- To simplify the creation of structural primitives, a grid or drawing plane is used. The drawing plane does not in any way limit the user to two dimensional coordinates but instead is used as a guide to simplify the creation of structural primitives. The drawing plane is represented by the active grid plane (The grid does not have to be visible). To demonstrate how drawing planes are used, review the following section: **Creating and Viewing Simple Structures**.

Active Cursor

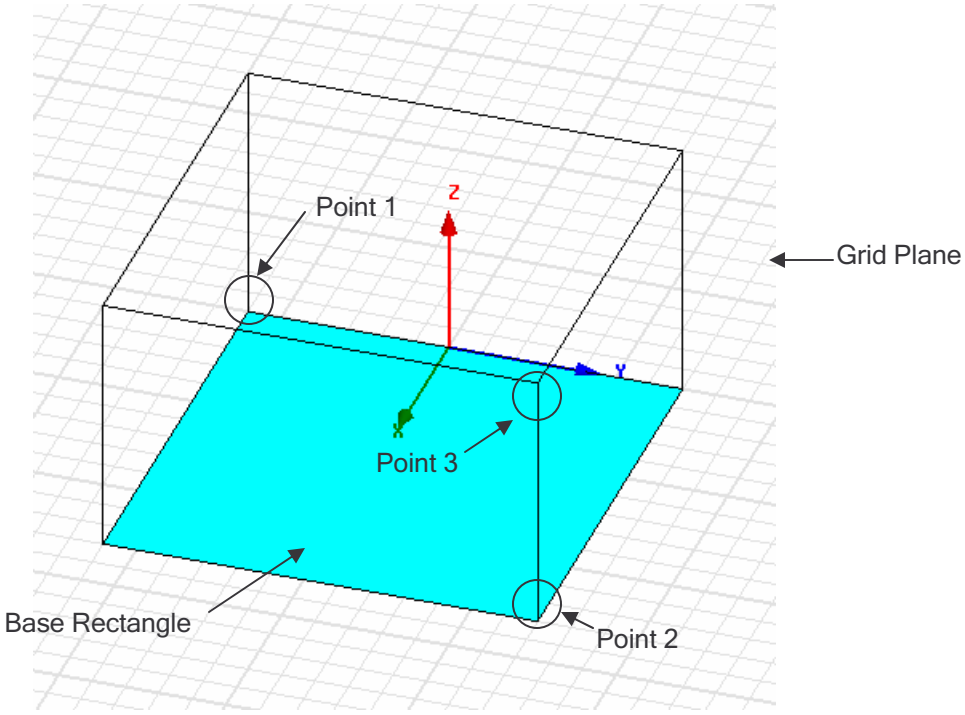
- The active cursor refers to the cursor that is available during object creation. The cursor allows you to graphically change the current position. The position is displayed on the status bar of the Ansoft HFSS Desktop Window.

- When objects are not being constructed, the cursor remains passive and is set for dynamic selection. See the Overview of Selecting Objects for more details.

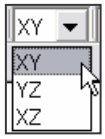
Creating and Viewing a Simple Structure

- Creating 3D structural objects is accomplished by performing the following steps:
  1. Set the grid plane
  2. Create the base shape of the object
  3. Set the Height

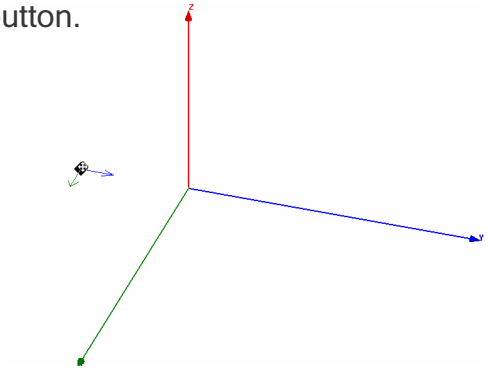
- **Create a Box**
  - We will investigate creating a box to demonstrate these steps. These steps assume that project and a HFSS design have already been created. Three points are required to create the box. The first two form the base rectangle and the third sets the height.
    - Point 1: Defines the start point of the base rectangle
    - Point 2: Defines the size of the base rectangle
    - Point 3: Defines the height of the Box



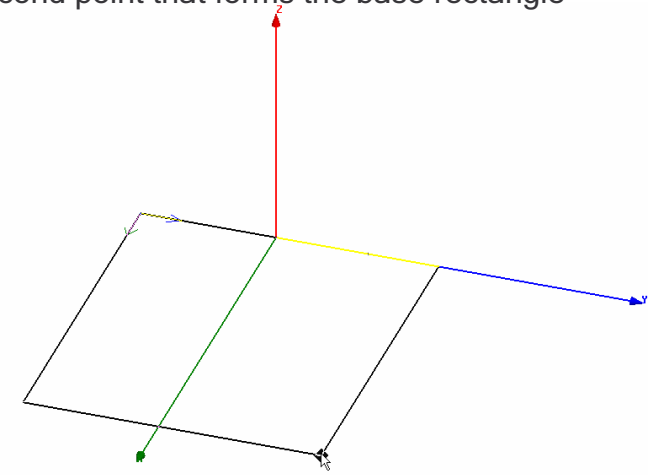
- **Create a Box (Continued)**



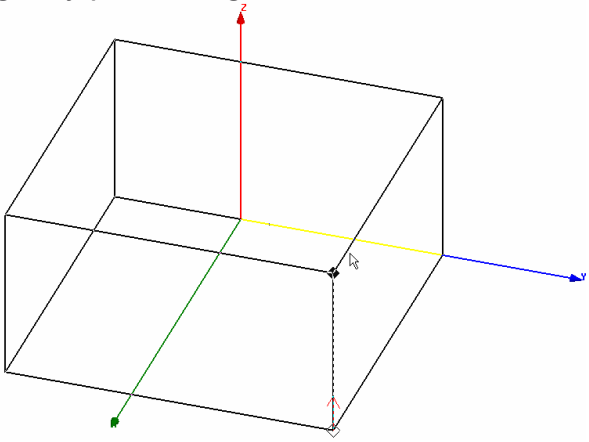
1. Select the menu item ***3D Modeler > Grid Plane > XY***
2. Use the mouse to create the base shape
    1. Set the start point by positioning the active cursor and click the left mouse button.



    2. Position the active cursor and click the left mouse button to set the second point that forms the base rectangle



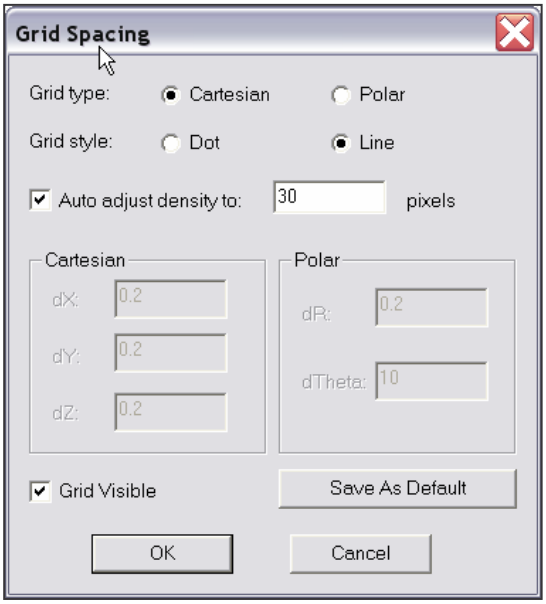
3. Set the Height by positioning the active cursor and clicking left mouse button.



Specifying Points

Grid

- From the example, we saw that the simplest way to set a point is by clicking its position on the grid plane. To set the precision of the grid plane, select the menu item ***View > Grid Settings.*** From here you may specify the Grid Type, Style, Visibility, and Precision. By pressing the **Save As Default** button, you can set the default behavior for future HFSS Designs.

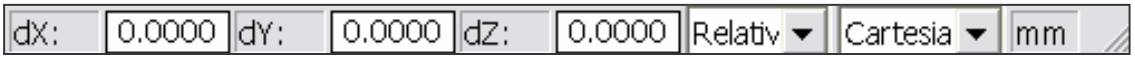


Coordinate Entry

- Another way to specify a coordinate is to use the Coordinate Entry fields which are located on the status bar of the Ansoft HFSS Desktop. The position may be specified in **Cartesian, Cylindrical**, or **Spherical** coordinates. Once the first point is set, the Coordinate Entry will default to Relative coordinates. In Relative mode the coordinates are no longer absolute (measured from the origin of the working coordinate system), but relative to the last point entered.

Equations

- The Coordinate Entry fields allow equations to be entered for position values. Examples: 2*5, 2+6+8, 2*cos(10*(pi/180)).
- Variables are not allowed in the Coordinate Entry Field
- **Note**: Trig functions are in radians

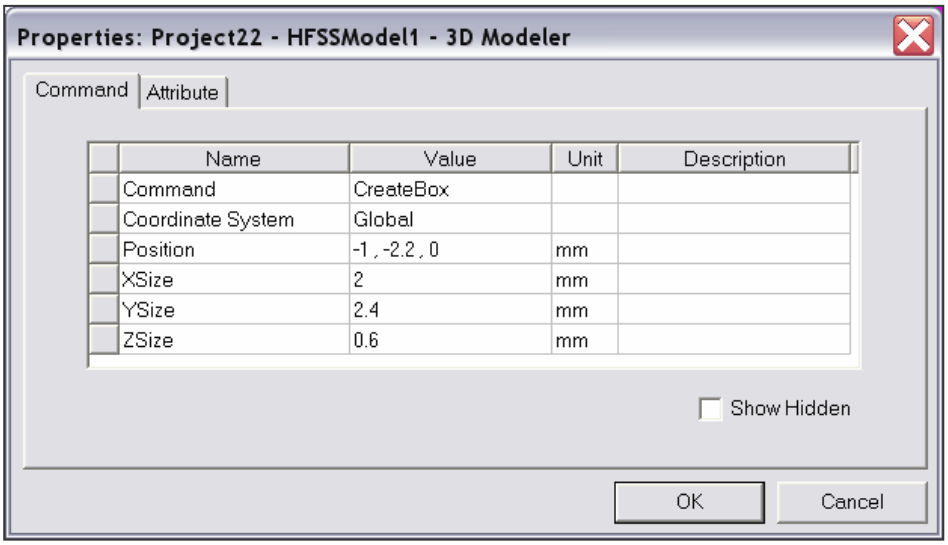


Relative mode

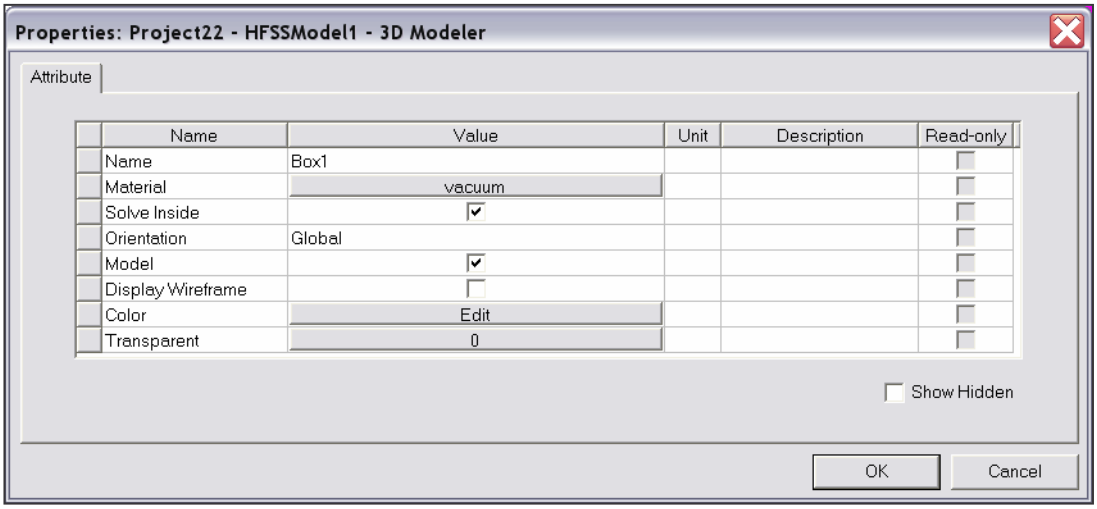
Specifying Points (Continued)

- **Object Properties**
  - By default the Properties dialog will appear after you have finished sketching an object. The position and size of objects can be modified from the dialog. This method allows you to create objects by clicking the estimated values using the mouse and then correcting the values in the final dialog.
  - The Property dialog accepts equations, variables, and units. See the **Overview of Entering Parameters** for more detail.
  - Every object has two types of properties
    1. **Command** – Defines the structural primitive
    2. **Attributes** – Defines the material, display, and solve properties

Commands



Attributes



Overview of Draw

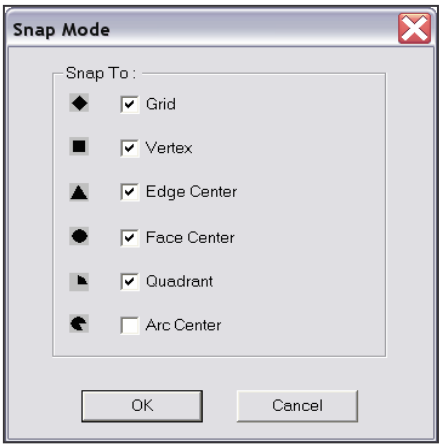
- **Primitives**
  - In solid modeling, the basic element or object is often called a primitive. Examples of primitives are boxes, cylinders, rectangles, circles, etc. There are two types of primitives: 3D primitives or solids, and 2D primitives or surfaces. By placing a collection of primitives in the correct location and of the correct size we can create a represent complex structural objects.
  - To create complex objects, primitives can be used as “tools” to cut holes, carve away, or join. The operations that are performed with these “tools” are often referred to as Boolean operations.
  - 2D primitives can be swept to create arbitrarily shaped solid primitives

- **2D Draw Objects**
  - The following 2D Draw objects are available:
    - Rectangle, Circle, Line, Point, Spline, Ellipse, Regular Polygon (v8.5 circle)

- **3D Draw Objects**
  - The following 3D Draw objects are available:
    - Box, Cylinder, Sphere, Torus, Helix, Bond Wire, Cone, Regular Polyhedron (v8.5 cylinder)

- **True Surfaces**
  - Circles, Cylinders, Spheres, etc are represented as true surfaces. In versions prior to release 9, these primitives would be represented as faceted objects. If you wish to use the faceted primitives (Cylinders or Circles), select the Regular Polyhedron or Regular Polygon.
  - To control the mesh generation of true surfaces objects, see the section on Mesh Control.

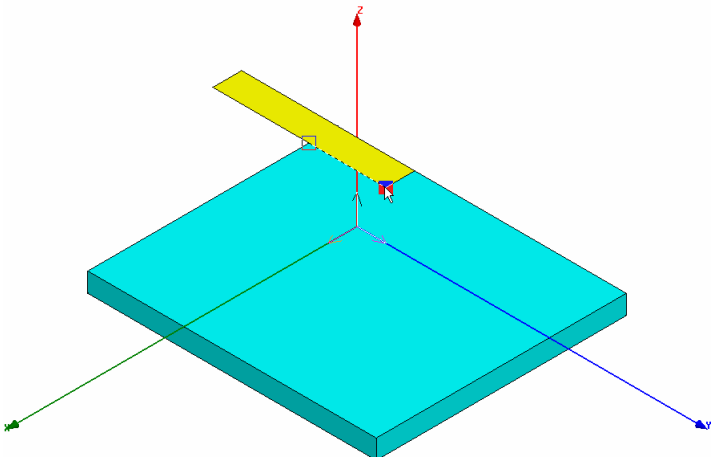
Overview of Draw (Continued)



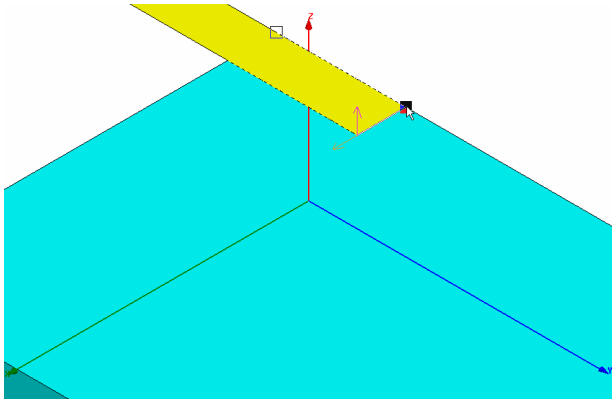
- **Snap Mode**
  - As an aid for graphical selection, the modeler provides Snap options. The default is to snaps are shown here. The shape of the active cursor will dynamically change as the cursor is moved over the snap positions.

- **Moving**
  - By default all active cursor movement is in three dimensions. The modeler can also be set to allow the active cursor to only move in a plane or out of plane. These are set from the menu item **3D Modeler > Movement Mode.**
  - In addition, the movement can be limited to a specific direction (x, y, or z) by holding down the x, y, or z key. This prevents movement in the other directions.
  - Pressing the **CTRL+Enter** key sets a local reference point. This can be useful for creating geometry graphically that is based on an existing objects. This is outlined on the next page:

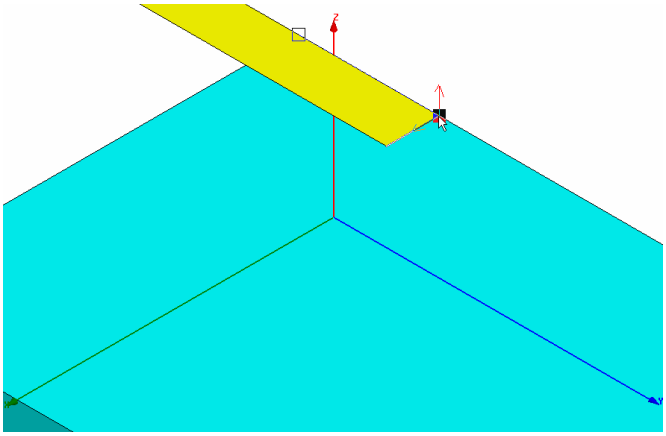
Moving (Continued)



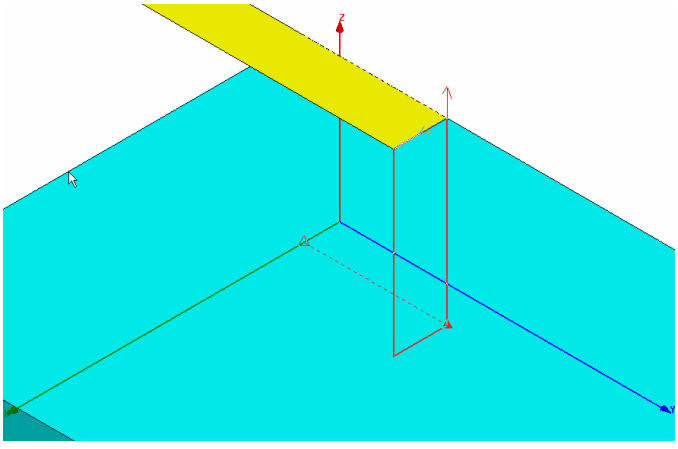
Step 1: Start Point



Step 2: Hold X key and select vertex point



Step 3: CTRL+Enter Keys set a local reference



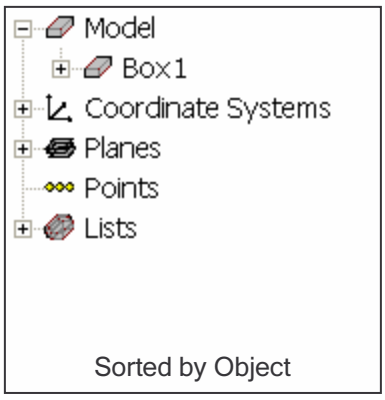
Step 4: Hold Z key and set height

Selecting Previously Defined Shapes

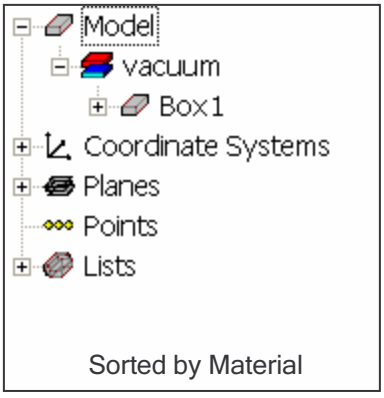
- You may select an object by moving the mouse over the object in the graphics area and clicking on it. The default mode is Dynamic selection which will display the object to be selected with a unique outline color. Please note that after selecting (Clicking on the object) the object it will be displayed solid pink while all other objects are drawn transparent.
- **Types of Selection**
  - The default is to select objects. Sometimes is necessary to select faces, edges, or vertices. To change the selection mode, select the menu item ***Edit > Select*** and choose the appropriate selection mode. The shortcut keys **o** (Object selection) and **f** (face selection) are useful for quickly switching between the most common selection modes
- **Multiple Select or Toggle Selection**
  - Multiple objects can be selected graphically by holding down the CTRL key while selecting. In addition, with the CTRL key pressed, the selection of an object can be toggled between selected or unselected.
- **Blocked Objects**
  - If the object you wish to select is located behind another object, select the object that is blocking the desired object and press the **b** key or right-click and select **Next Behind** from the context menu. You may repeat this as many times as needed to select the correct object.
- **Select All Visible**
  - You can select all visible objects by pressing the **CTRL+a** key or by selecting the menu item ***Edit > Select All Visible.***
- **Select by Name**
  - To select objects by Name you can use anyone of the following:
    - Select the menu item ***Edit > Select > By Name***
    - Select the menu item ***HFSS > List***
      - Select the **Model** tab
      - Select objects from the list
    - Use the ***Model Tree***. See the next page

Selecting Previously Defined Shapes (Continued)

- **Model Tree**
  - After an object has been created, it is automatically added to the Model Tree. All objects can be found in the Model Tree. If you open the Model folder you will find the objects sorted by Object or by Material. You can toggle between the views by toggling the menu item ***3D Modeler > Group Object by Material***.

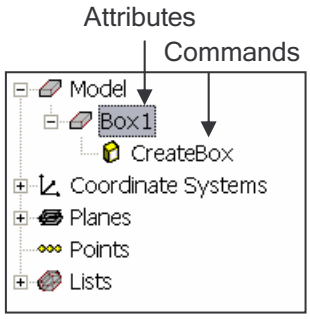


Sorted by Object



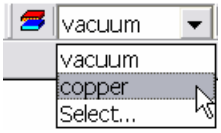
Sorted by Material

  - As stated previously, every object has two types of properties:
    - **Attributes**
      - You may select an object by clicking on the corresponding item in the Model Tree.
      - When the object is selected the attributes will be displayed in the Property Window. Double-clicking on the object will open a properties dialog. Use the Property Window or properties dialog to modify the attributes.
    - **Commands**
      - From the Model Tree, the Command Properties can be selected by expanding the object folder to display the command list. Using the mouse, select the corresponding command from the tree. The properties will be displayed in the Property Window. Double-clicking on the command will open a properties dialog. Use the Property Window or properties dialog to modify the command.
      - When the command is selected, the object will be outlined with bold lines in the 3D Model Window. Since an object can be a combination of several primitives, the command list may contain several objects. Anyone of these commands can be selected to visualize or modify the object.

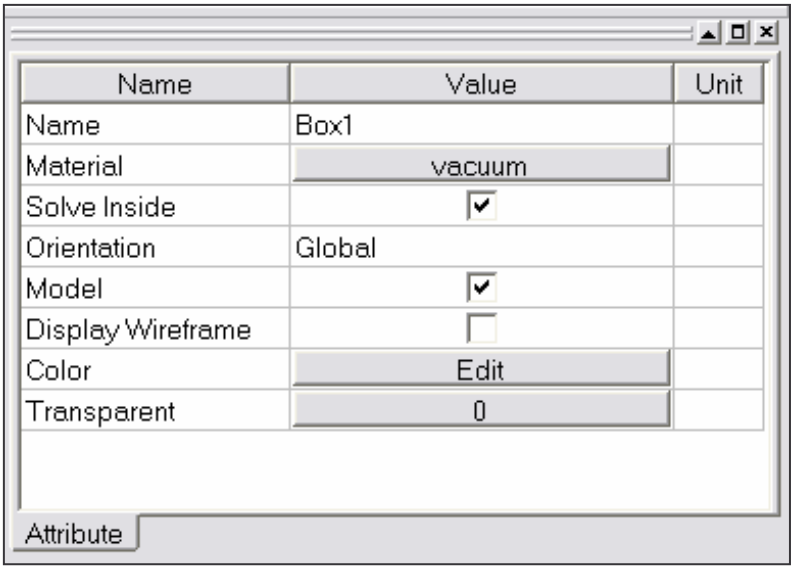


Object Attributes

- An objects attributes set the following user defined properties:
  - **Name** – User defined name. Default names start with the primitive type followed by an increasing number: Box1, Box2, etc.
  - **Material** – User defined material property. The default property is vacuum. This can be changed by using the material toolbar

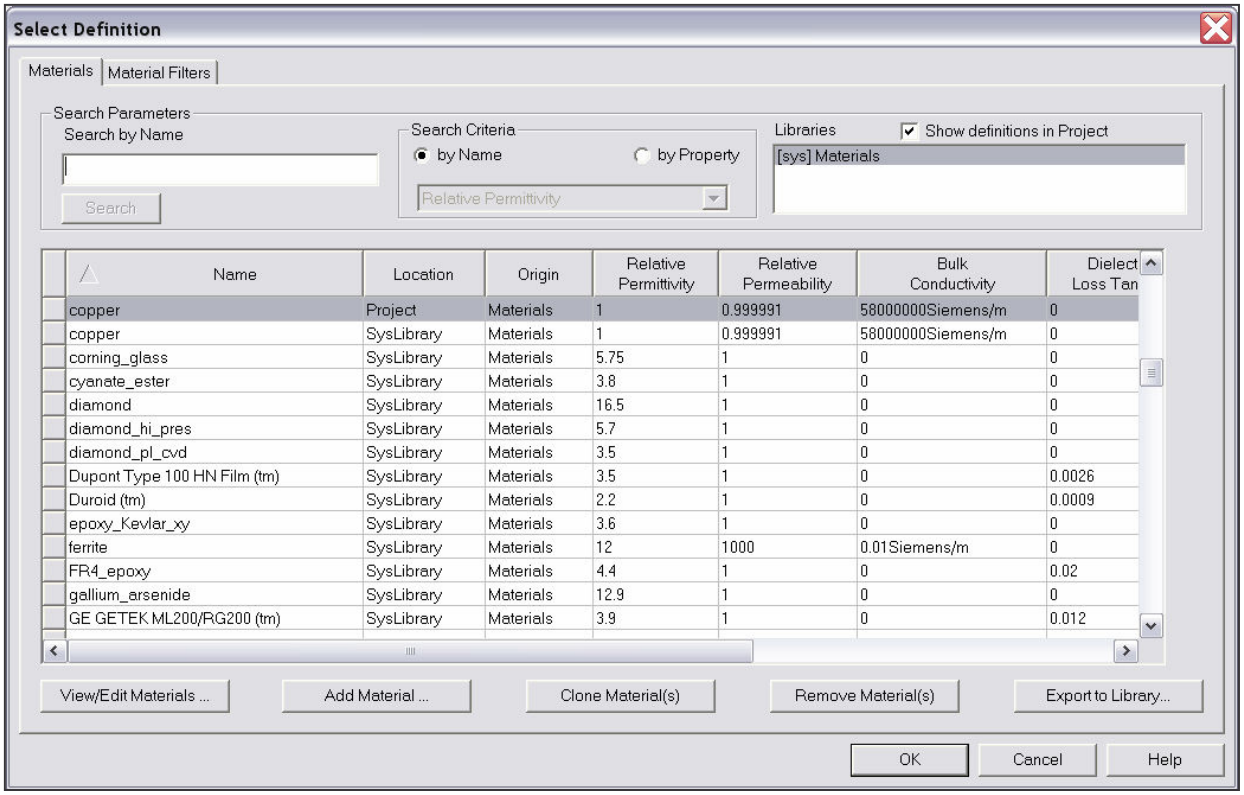


  - **Solve Inside** – By default HFSS only solves for fields inside dielectrics. To force HFSS to solve inside conductors, check solve inside.
  - Orientation
  - **Model Object** – Controls if the object is included in the solve
  - **Display Wireframe** – Forces the object to always be displayed as wireframe
  - **Color –** Set object color
  - **Transparency –** Set the transparency of an object. 0-Solid, 1- Wireframe
- **Note:** Visibility is not an object property.



Materials

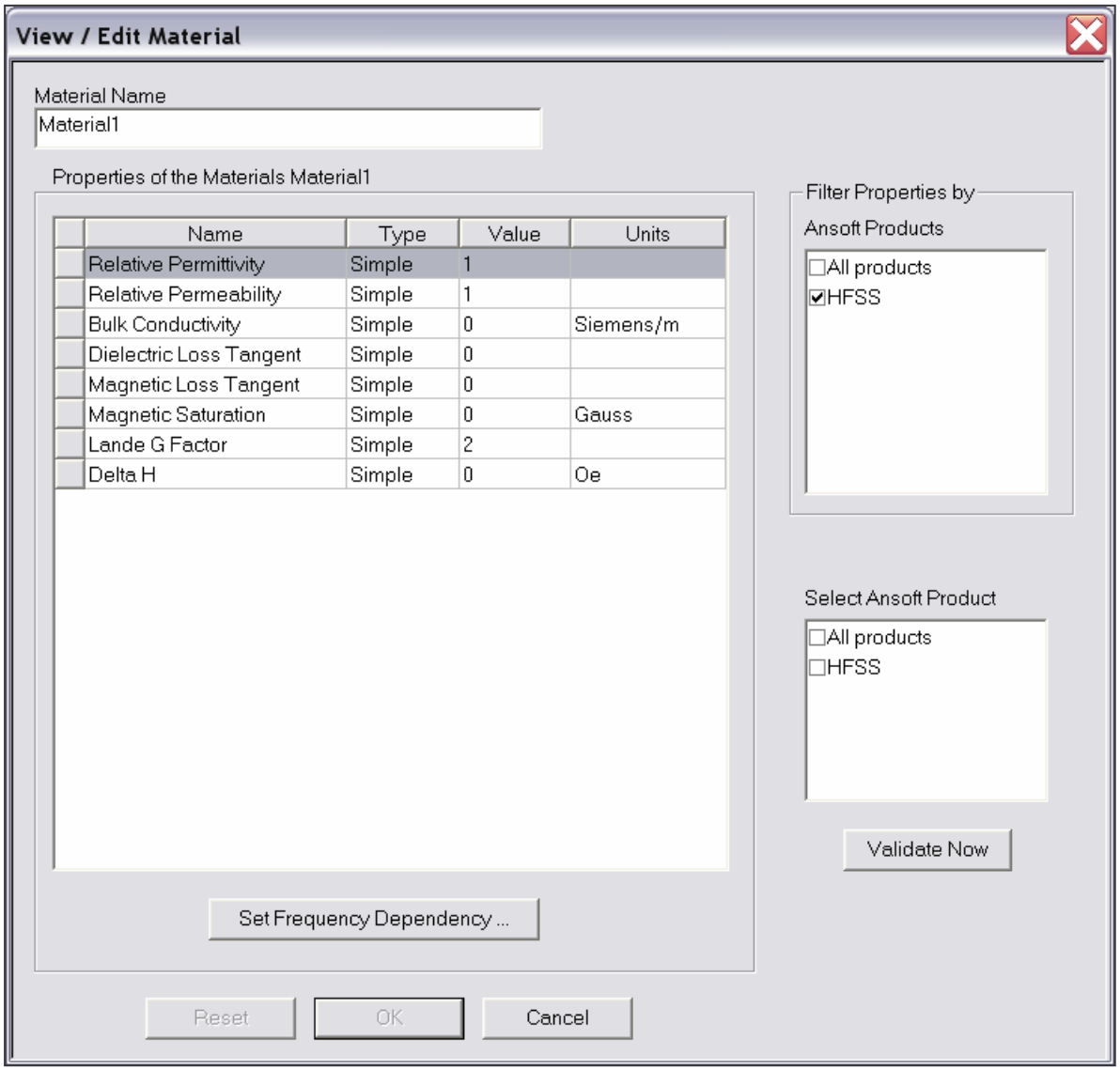
- By clicking on the property button for the material name, the material definition window will appear. You can select from the existing database or define a custom project material.



Materials (Continued)

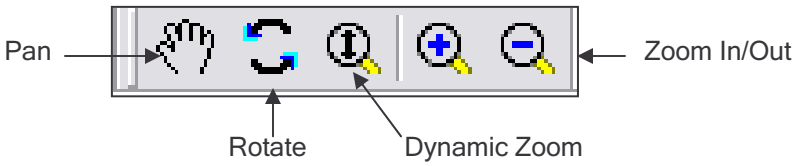
User Defined Project Material

- To define a custom material click the **Add Material** button from the material definition window. The following dialog will appear. Enter the material definitions and click the OK button.

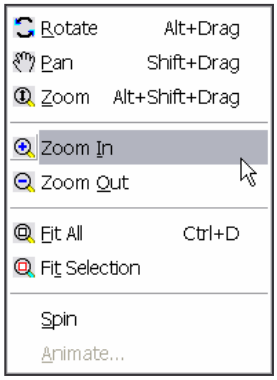


Changing the View

- You can change the view at any time (even during shape generation) by using the following commands:
- **Toolbar**
  - **Rotate** – The structure will be rotated around the coordinate system
  - **Pan** – The structure will be translated in the graphical area
  - **Dynamic Zoom** – Moving the mouse upwards will increase the zoom factor while moving the mouse downwards will decrease the zoom factor
  - **Zoom In/Out** – In this mode a rubber band rectangle will be defined by dragging the mouse. After releasing the mouse button the zoom factor will be applied.



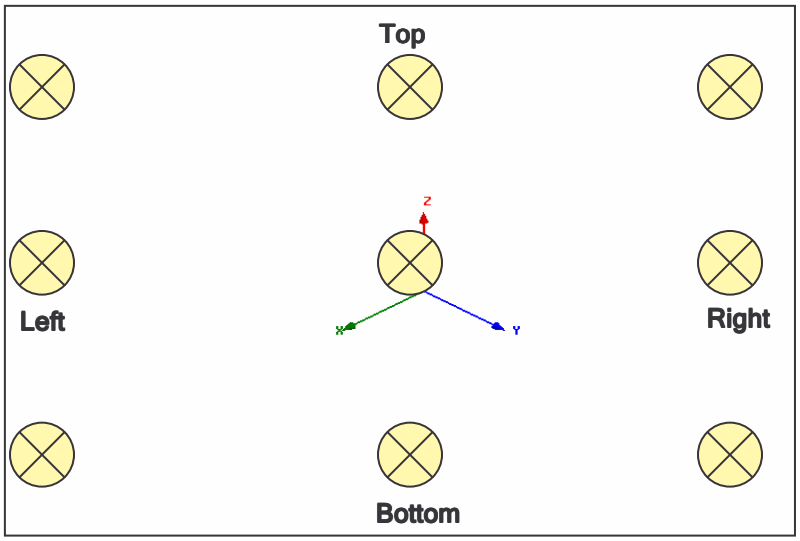
- **Context Menu**
  - Right click in the graphics area and select the menu item **View** and choose from the options outlined in the Toolbar section. The context menu also offers the following:



    - **Fit All** – This will zoom the defined structure to a point where it fits in the drawing area
    - **Fit Selection** – This fits only the selected objects into the drawing area.
    - **Spin** – Drag the mouse and release the mouse button to start the object spinning. The speed of the dragging prior to releasing the mouse controls the speed of the spin.
    - **Animate** – Create or display the animation of parametric geometry

- **Shortcuts**
  - Since changing the view is a frequently used operation, some useful shortcut keys exist. Press the appropriate keys and drag the mouse with the left button pressed:
    - **ALT + Drag** – Rotate
      - In addition, there are 9 pre-defined view angles that can be selected by holding the ALT key and double clicking on the locations shown on the next page.
    - **Shift + Drag** – Pan
    - **ALT + Shift + Drag** – Dynamic Zoom

Shortcuts - Predefined Views



Predefined View Angles

Changing the View (Continued)

- **Visibility**
  - The visibility of objects, Boundaries, Excitations, and Field Reports can be controlled from the menu item ***View > Visibility***
- **Hide Selection**
  - The visibility of selected objects can be set hidden by selecting the object(s) and choosing the menu ***View > Hide Selection > All Views***.
- **Rendering**
  - To change the rendering select the menu item View > Render > Wireframe or View > Render > Smooth Shaded
- **Coordinate System**
  - To control the view of the coordinate system, select the menu item:
    - **Visibility:**
      - Toggle the menu item ***View > Coordinate System > Hide (Show)***
    - **Size**:
      - Toggle the menu item ***View > Coordinate System > Small (Large)***
- **Background Color**
  - To set the background color, select the menu item **View > Modify Attributes > Background Color**
- **Addition View Seetings**
  - Additional attributes of the view such as the projection, orientation, and lighting can be set from the menu item **View > Modify Attributes**

Applying Structural Transformations

- So far we have investigated hot to model simple shapes and how to change the view of the model. To create more complicated models or reduce the number of objects that need to be created manually we can apply various transformations.
- The following examples assume that you have already selected the object(s) that you wish to apply a transformation.
- You can select the transformation options from the menu item ***Edit >***
  - ***Arrange >***
    - **Move –** Translates the structure along a vector
    - **Rotate –** Rotates the shape around a coordinate axis by an angle
    - **Mirror –** Mirrors the shape around a specified plane
    - **Offset –** Performs a uniform scale in x, y, and z.
  - ***Duplicate >***
    - **Along Lines –** Create multiple copies of an object along a vector
    - **Around Axis –** Create multiple copies of an object rotated by a fixed angle around the x, y, or z axis
    - **Mirror -** Mirrors the shape around a specified plane and creates a duplicate
  - ***Scale*** – Allows non-uniform scaling in the x, y, or z direction
- The faces of an object can also be moved to alter the shape of an existing object. To move the faces of an object select the menu item ***3D Modeler > Surfaces > Move Faces*** and select ***Along Normal*** or ***Along Vector***.

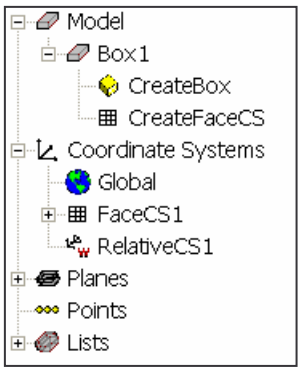
Combine Objects by Using Boolean Operations

- Most complex structures can be reduced to combinations of simple primitives. Even the solid primitives can be reduced to simple 2D primitives that are swept along a vector or around an axis(Box is a square that is swept along a vector to give it thickness). The solid modeler supports the following Boolean operations:
  - **Unite** – combine multiple primitives
    - Unite disjoint objects
      - Separate Bodies to separate
  - **Subtract** – remove part of a primitive from another
  - **Split** – break primitives into multiple parts
  - **Intersect**– keep only the parts of primitives that overlap
  - **Sweep** – turn a 2D primitive into a solid by sweeping: Along a Vector, Around an Axis, Along a Path
  - **Connect** – connect 2D primitives. Use Cover Surfaces to turn the connected object into a solid
  - **Section** – generate 2D cross-sections of a 3D object
- Most Boolean operations require a base primitive in which the Boolean operation is performed. Only the base object will be preserved.
- The Boolean functions provide the option to Clone objects.

Local Coordinate Systems

- The ability to create local coordinate systems adds a great deal of flexibility to the creations of structural objects. In previous sections we have only discussed objects that are aligned to the global coordinate system. The local coordinate system simplifies the definition of objects that do not align with the global coordinate system. In addition, the object history is defined relative to a coordinate system. If the coordinate system is moved, the geometry will automatically move with it. The definition of coordinate systems are maintained in the Model Tree.

- **Working Coordinate System**
  - The working coordinate system is the currently selected CS. This can be a local or global CS



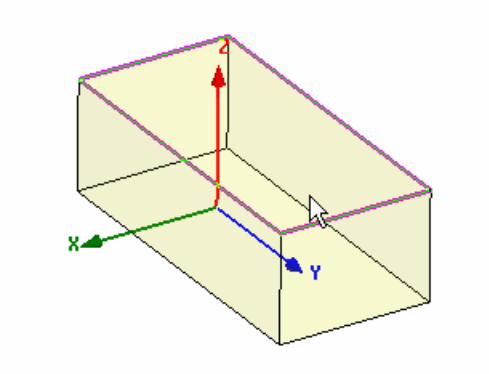
- **Global CS**
  - The default fixed coordinate system

- **Relative CS**
  - User defined local coordinate system.
    - Offset
    - Rotated
    - Both

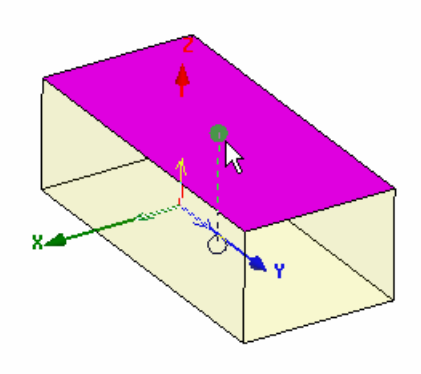
- **Face CS**
  - User defined local coordinate system. It is tied to the location of the object face it was created on. If the size of the base object changes, all objects created relative to the face CS will be updated automatically.
  - Continued on Next Page

Local Coordinate Systems (Continued)

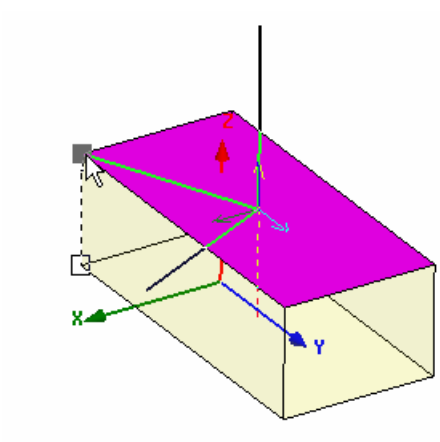
- **Face CS (Continued)**
  - To create a face CS, select the menu item ***3D Modeler > Coordinate System > Face***
    1. Graphically select Face (Highlighted in model)
    2. Select Origin for Face CS
    3. Set X-Axis



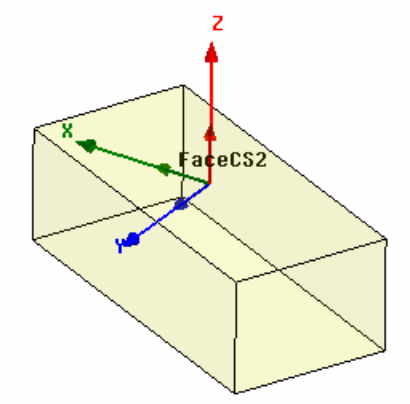
Step 1: Select Face



Step 2: Select Origin



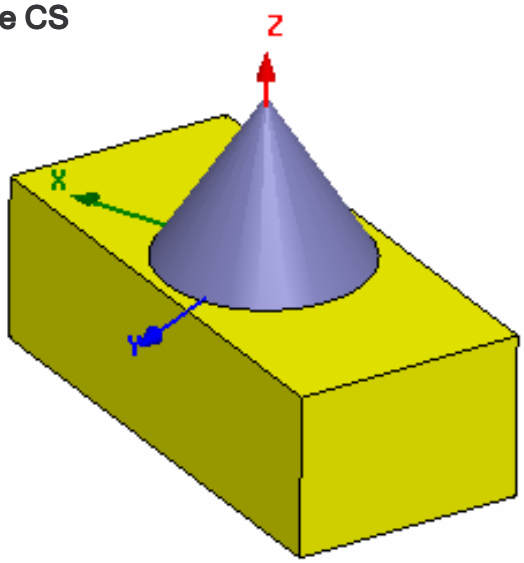
Step 3: Set X-Axis



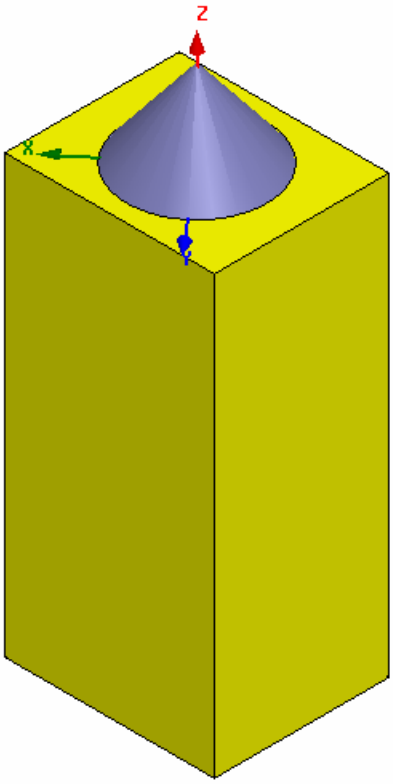
New Working CS

Local Coordinate Systems (Continued)

Example of Face CS



Cone is created with Face CS

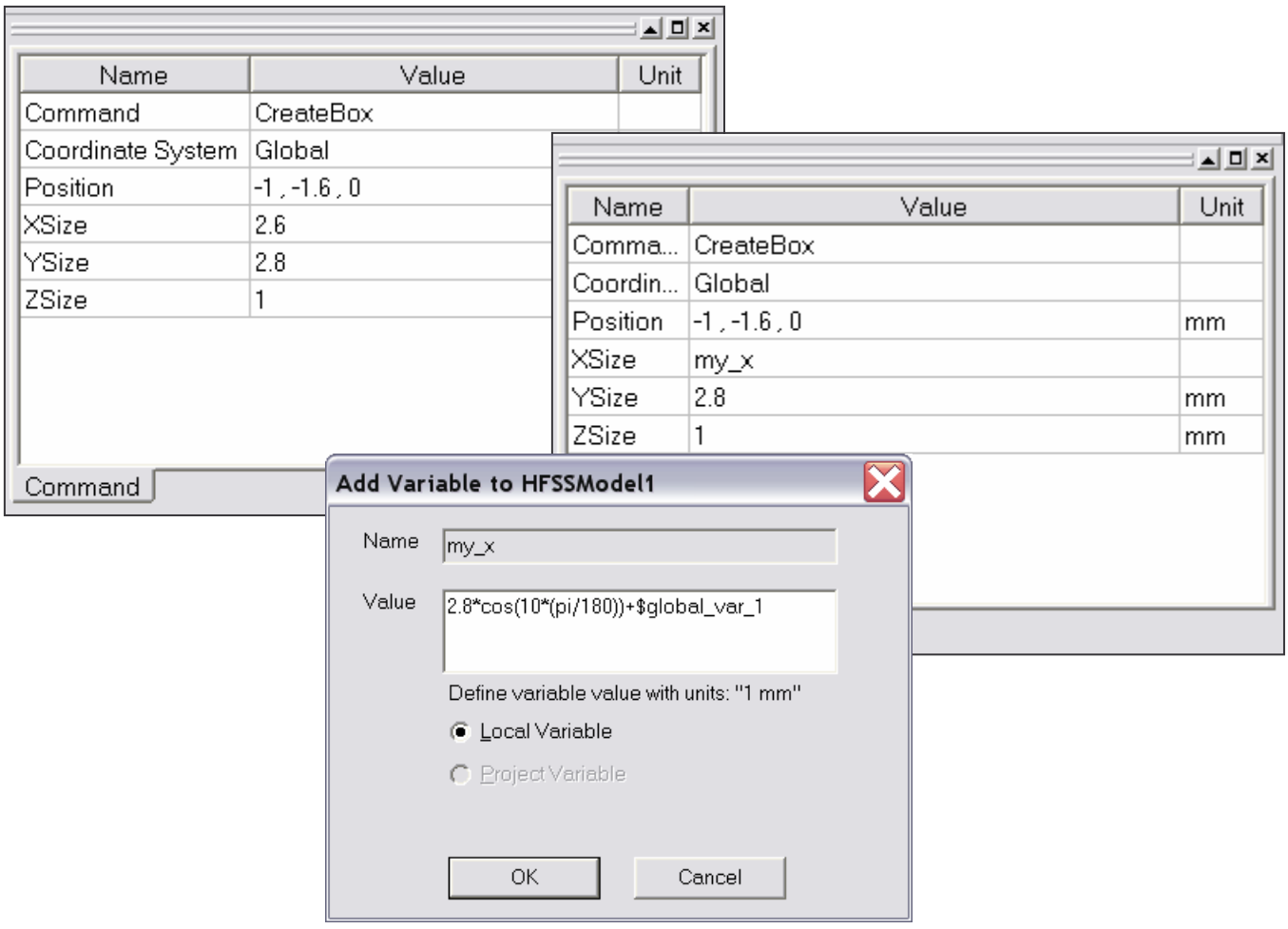


Change the size of the box and the Cone is automatically moved with the Face CS

Parametric Geometry

- The parametric modeler capability allows us to define variables in replace of a fixed position or size. Once this has been defined the variable can be changed by the user or by Optimetrics. Optimetrics can then be used to perform automatic Optimization, Parametric Sweeps, Statistical, or Sensitivity Analysis.

- **Defining Parameters**
  - Select the command to parameterized
  - Choose the value to change
  - Enter a variable in replace of the fixed value
  - Define the variable using any combination of math functions or design variables.
  - The model will automatically be updated



Parametric Geometry (Continued)

- **Variables**
  - There are two types of variables that can be defined in the HFSS Desktop
    - **Design Properties** – Local to model. To access the local variables select the menu item ***HFSS > Design Properties***
    - **Project Variables** – Global to all models in project. Start with $. To access the global or project variables, select the menu item ***Project > Project Variables***
  - **Units**
    - When defining variables they must contain units. The default units for variables is meters.
  - **Equations**
    - The variables can contain complex equations. See the Online Help for a complete list of math functions
  - **Animation**
    - Right-Click in the 3D Model Window & Choose Animate to preview the parameterization

This page is intentionally left blank.

Boundary Conditions

- This chapter describes the basics for applying boundary conditions. Boundary conditions enable you to control the characteristics of planes, faces, or interfaces between objects. Boundary conditions are important to understand and are fundamental to solution of Maxwell's equations.

Why they are Important

- The wave equation that is solved by Ansoft HFSS is derived from the differential form of Maxwell's Equations. For these expressions to be valid, it is assumed that the field vectors are single-valued, bounded, and have continuous distribution along with their derivatives. Along boundaries or sources, the fields are discontinuous and the derivatives have no meaning. Therefore boundary conditions define the field behavior across discontinuous boundaries.
- As a user of Ansoft HFSS you should be aware of the field assumptions made by boundary conditions. Since boundary conditions force a field behavior we want to be aware of the assumptions so we can determine if they are appropriate for the simulation. Improper use of boundary conditions may lead to inconsistent results.
- When used properly, boundary conditions can be successfully utilized to reduce the model complexity. In fact, Ansoft HFSS automatically uses boundary conditions to reduce the complexity of the model. Ansoft HFSS can be thought of as a virtual prototyping world for passive RF devices. Unlike the real world which is bounded by infinite space, the virtual prototyping world needs to be made finite. In order to achieve this finite space, Ansoft HFSS applies a background or outer boundary condition which is applied to the region surrounding the geometric model.
- The model complexity usually is directly tied to the solution time and computer resources so it is a competitive advantage to utilize them whenever possible.

Common Boundary Conditions

- There are three types of boundary conditions. The first two are largely the users responsibility to define them or ensure that they are defined correctly. The material boundary conditions are transparent to the user.

1. **Excitations**
   - Wave Ports (External)
   - Lumped Ports (Internal)
2. **Surface Approximations**
   - Symmetry Planes
   - Perfect Electric or Magnetic Surfaces
   - Radiation Surfaces
   - Background or Outer Surface
3. **Material Properties**
   - Boundary between two dielectrics
   - Finite Conductivity of a conductor

How the Background Affects a Structure

- The background is the region that surrounds the geometric model and fills any space that is not occupied by an object. Any object surface that touches the background is automatically defined to be a Perfect E boundary and given the boundary name *outer*. You can think of your structure as being encased with a thin, perfect conductor.

- If it is necessary, you can change a surface that is exposed to the background to have properties that are different from *outer*:
  - To model losses in a surface, you can redefine the surface to be either a Finite Conductivity or Impedance boundary. A Finite Conductivity boundary can be a lossy metal, with loss as a function of frequency and defined using conductivity and relative permeability parameters. An Impedance boundary has real or complex values that by default remain constant over frequency.
  - To model a surface to allow waves to radiate infinitely far into space, redefine the surface to be radiation boundary.

- The background can affect how you make material assignments. For example, if you are modeling a simple air-filled rectangular waveguide, you can create a single object in the shape of the waveguide and define it to have the characteristics of air. The surface of the waveguide is automatically assumed to be a perfect conductor and given the boundary condition *outer*, or you can change it to a lossy conductor.

Technical Definition of Boundary Conditions

- **Excitation** – An excitation port is a type of boundary condition that permits energy to flow into and out of a structure. See the section on Excitations.

- **Perfect E** – Perfect E is a perfect electrical conductor, also referred to as a perfect conductor. This type of boundary forces the electric field (E-Field) perpendicular to the surface. There are also two automatic Perfect E assignments:
  - Any object surface that touches the background is automatically defined to be a Perfect E boundary and given the boundary condition name *outer*.
  - Any object that is assigned the material pec (Perfect Electric Conductor) is automatically assigned the boundary condition Perfect E to its surface and given the boundary condition name *smetal.*

- **Perfect H** – Perfect H is a perfect magnetic conductor. Forces E-Field tangential to the surface.
  - **Natural** – for a Perfect H boundary that overlaps with a perfect E boundary, this reverts the selected area to its original material, erasing the Perfect E boundary condition. It does not affect any material assignments. It can be used, for example, to model a cut-out in a ground plane for a coax feed.

- **Finite Conductivity** –A Finite Conductivity boundary enables you to define the surface of an object as a lossy (imperfect) conductor. It is an imperfect E boundary condition, and is analogous to the lossy metal material definition. To model a lossy surface, you provide loss in Siemens/meter and permeability parameters. Loss is calculated as a function of frequency. It is only valid for good conductors. Forces the tangential E-Field equal to $Z_s(n \times H_{tan})$. The surface impedance (Z_s) is equal to, $(1+j)/(\delta\sigma)$, where:
  - δ is the skin depth, $(2/(\omega\sigma\mu))^{0.5}$ of the conductor being modeled
  - ω is the frequency of the excitation wave.
  - σ is the conductivity of the conductor
  - μ is the permeability of the conductor

Technical Definition of Boundary Conditions (Continued)

- **Impedance** – a resistive surface that calculates the field behavior and losses using analytical formulas. Forces the tangential E-Field equal to $Z_s(n \times H_{tan})$. The surface impedance is equal to $R_s + jX_s$, where:
  - Rs is the resistance in ohms/square
  - Xs is the reactance in ohms/square

- **Layered Impedance** – Multiple thin layers in a structure can be modeled as an impedance surface. See the Online Help for additional information on how to use the Layered Impedance boundary.

- **Lumped RLC** – a parallel combination of lumped resistor, inductor, and/or capacitor surface. The simulation is similar to the **Impedance** boundary, but the software calculate the ohms/square using the user supplied R, L, C values.

- **Infinite Ground Plane** – Generally, the ground plane is treated as an infinite, Perfect E, Finite Conductivity, or Impedance boundary condition. If radiation boundaries are used in a structure, the ground plane acts as a shield for far-field energy, preventing waves from propagating past the ground plane. to simulate the effect of an infinite ground plane, check the Infinite ground plane box when defining a Perfect E, Finite Conductivity, or Impedance boundary condition.

- **Radiation** – Radiation boundaries, also referred to as absorbing boundaries, enable you to model a surface as electrically open: waves can then radiate out of the structure and toward the radiation boundary. The system absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure and into space. Radiation boundaries may also be placed relatively close to a structure and can be arbitrarily shaped. This condition eliminates the need for a spherical boundary. For structures that include radiation boundaries, calculated S-parameters include the effects of radiation loss. When a radiation boundary is included in a structure, far-field calculations are performed as part of the simulation.

This page is intentionally left blank.

Technical Overview

- Ports are a unique type of boundary condition that allow energy to flow into and out of a structure. You can assign a port to any 2D object or 3D object face. Before the full three-dimensional electromagnetic field inside a structure can be calculated, it is necessary to determine the excitation field pattern at each port. Ansoft HFSS uses an arbitrary port solver to calculate the natural field patterns or modes that can exist inside a transmission structure with the same cross section as the port. The resulting 2D field patterns serve as boundary conditions for the full three-dimensional problem.

  - By default Ansoft HFSS assumes that all structures are completely encased in a conductive shield with no energy propagating through it. You apply **Wave Ports** to the structure to indicate the area were the energy enters and exits the conductive shield.

  - As an alternative to using **Wave Ports**, you can apply **Lumped Ports** to a structure instead. **Lumped Ports** are useful for modeling internal ports within a structure.

Wave Port

- The port solver assumes that the **Wave Port** you define is connected to a semi-infinitely long waveguide that has the same cross-section and material properties as the port. Each **Wave Port** is excited individually and each mode incident on a port contains one watt of time-averaged power. **Wave Ports** calculate characteristic impedance, complex propagation constant, and generalized S-Parameters.

Wave Equation

- The field pattern of a traveling wave inside a waveguide can be determined by solving Maxwell's equations. The following equation that is solved by the 2D solver is derived directly from Maxwell's equation.

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times E(x,y) \right) - k_0^2 \varepsilon_r E(x,y) = 0$$

- where:
  - **E**(x,y) is a phasor representing an oscillating electric field.
  - k_0 is the free space wave number,
  - μ_r is the complex relative permeability.
  - ε_r is the complex relative permittivity.

- To solve this equation, the 2D solver obtains an excitation field pattern in the form of a phasor solution, **E**(x,y). These phasor solutions are independent of *z* and *t*; only after being multiplied by $e^{-\gamma z}$ do they become traveling waves.

- Also note that the excitation field pattern computed is valid only at a single frequency. A different excitation field pattern is computed for each frequency point of interest.

Modes

- For a waveguide or transmission line with a given cross section, there is a series of basic field patterns (modes) that satisfy Maxwell's Equations at a specific frequency. Any linear combination of these modes can exist in the waveguide.

Mode Conversion

- In some cases it is necessary to include the effects of higher-order modes because the structure acts as a mode converter. For example, if the mode 1 (dominant) field at one port is converted (as it passes through a structure) to a mode 2 field pattern at another, then it is necessary to obtain the S-parameters for the mode 2 field.

Modes, Reflections, and Propagation

- It is also possible for a 3D field solution generated by an excitation signal of one specific mode to contain reflections of higher-order modes which arise due to discontinuities in a high frequency structure. If these higher-order modes are reflected back to the excitation port or transmitted onto another port, the S-parameters associated with these modes should be calculated. If the higher-order mode decays before reaching any port–either because of attenuation due to losses or because it is a non-propagating evanescent mode–there is no need to obtain the S-parameters for that mode.

Modes and Frequency

- The field patterns associated with each mode generally vary with frequency. However, the propagation constants and impedances always vary with frequency. Therefore, when a frequency sweep has been requested, a solution is calculated for each frequency point of interest. When performing frequency sweeps, be aware that as the frequency increases, the likelihood of higher-order modes propagating also increases.

Modes and S-Parameters

- When the Wave Ports are defined correctly, for the modes that are included in the simulation, there is a perfect matched condition at the Wave Port. Because of this, the S-Parameters for each mode and Wave Port are normalized to a frequency dependent impedance. This type of S-Parameter is referred to as Generalized S-Parameter.

- Laboratory measurements, such as those from a vector network analyzer, or circuit simulators use a constant reference impedance (i.e. the ports are not perfectly matched at every frequency).

  - To obtain results consistent with measurements or for use with circuit simulators, the generalized s-parameters calculated by Ansoft HFSS must be renormalized to a constant characteristic impedance. See the section on **Calibrating Wave Ports** for details on how to perform the renormalization.

  - **Note:** Failure to renormalize the generalized S-Parameters may result in inconsistent results. For example, since the Wave Ports are perfectly matched at every frequency, the S-Parameters do not exhibit the interaction that actually exists between ports with a constant characteristic impedance.

Wave Port Boundary Condition

- The edge of a Wave Port can have the following boundary conditions:
  - Perfect E or Finite Conductivity – by default the outer edge of a Wave Port is defined to have a Perfect E boundary. With this assumption, the port is defined within a waveguide. For transmission line structures that are enclosed by metal, this is not a problem. For unbalanced or non-enclosed lines, the fields in the surrounding dielectric must be included. Improper sizing of the port definition will result in erroneous results.
  - Symmetry – the port solver understands Perfect E and Perfect H symmetry planes. The proper Wave Port impedance multiplier needs to be applied when using symmetry planes.
  - Impedance – the port solver will recognize an impedance boundary at the edges of the ports.
  - Radiation – the default setting for the interface between a Wave Port and a Radiation boundary is to apply a Perfect E boundary to the edge of the ports.

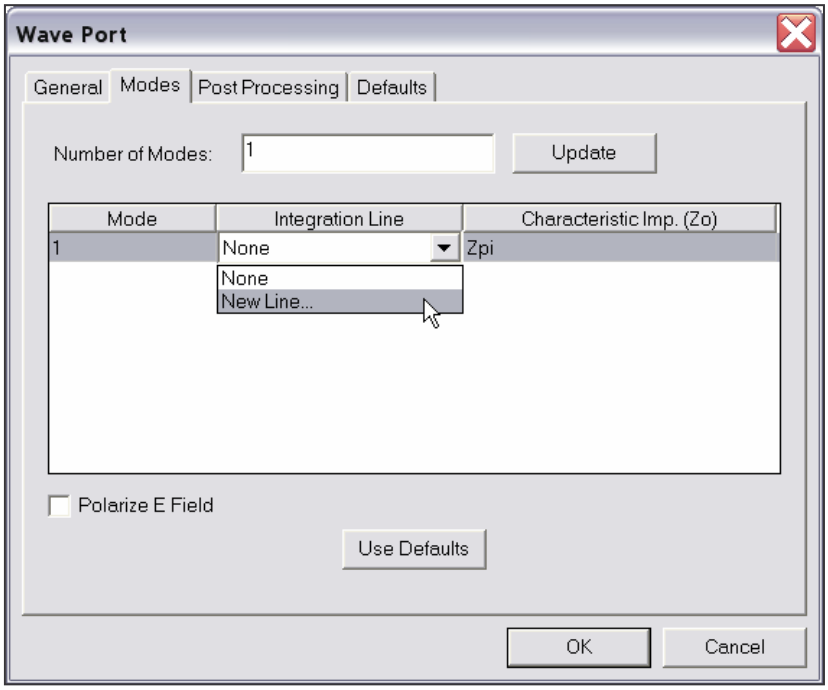
Calibrating Wave Ports

- Wave Ports that are added to a structure must be *calibrated* to ensure consistent results. This calibration is required in order to determine direction and polarity of fields and to make voltage calculations.

Solution Type: Driven Modal

- For Driven Modal simulations, the Wave Ports are calibrated using Integration Lines. Each Integration Line is used to calculate the following characteristics:
  - *Impedance* - As an impedance line, the line serves as the path over which Ansoft HFSS integrates the E-field to obtain the voltage at a Wave Port. Ansoft HFSS uses the voltage to compute the characteristic impedance of the Wave Ports, which is needed to renormalize generalized S-matrices to specific impedances such as 50 ohms.
    - **Note:** If you want to be able to renormalize S-parameters or view the values of Zpv or Zvi, you must apply Integration Lines to the Wave Ports of a structure.
  - *Calibration* - As a calibration line, the line explicitly defines the *up* or *positive* direction at each Wave Port. At any Wave Port, the direction of the field at $\omega t = 0$ can be in at least one of two directions. At some ports, such as circular ports, there can be more than two possible directions, and you will want to use Polarize E-Field. If you do not define an Integration Line, the resulting S-parameters can be out of phase with what you expect.
- **Tip** You may need to run a ports-only solution first to help determine how the Integration Lines need to be applied to a Wave Port and their direction.

- To calibrate a Wave Port, that has already been defined, with an Integration Line:
  1. From the **Project Tree**, expand **Excitations** and double click on the Wave Port to be calibrated
  2. Select the **Modes** tab.
  3. From the table, select the **Integration Line** column for the first mode and choose **New Line**.
  4. Enter the position and length of the line using one of the following methods
     - Type the start and stop points of the line directly into the x, y, or z axis fields, referenced to the working coordinates. For more information on coordinates, refer to “Getting Oriented in the Drawing Space” in Chapter **, Drawing Basics and Tips.
     - Graphically pick the points in the Design Window’s graphics area. The line is displayed as a vector; the vector indicates direction. From the **Integration Line** column, select **Swap Endpoints** to reverse the direction of the line, if necessary.
  5. Repeat steps 3 and 4 to define and apply lines to other modes of the current Wave Port.
  6. Click the **OK** button when you are finished defining Integration Lines
  7. Repeat steps 1-6 to apply lines to other Wave Ports.



Step 3: Create New Line

About Impedance Lines

- The S-matrices initially calculated by Ansoft HFSS are generalized S-matrices normalized to the impedances of each port. However, it is often desirable to compute S-matrices that are normalized to specific impedances such as 50 ohms. To convert a generalized S-matrix to a renormalized S-matrix, Ansoft HFSS first computes the characteristic impedance at each port. There are several ways to compute the characteristic impedance (Z_{pi}, Z_{pv}, Z_{vi}).

- Ansoft HFSS will always calculate $Z_{pi.}$ The impedance calculation using power and current is well defined for a Wave Port. The other two methods - Z_{pv} and Z_{vi} - require a line of integration to determine voltage. By defining an Integration Line for each mode, the voltage can be computed.

- In general, the impedance line should be defined between two points at which the voltage differential is expected to be a maximum. If you are analyzing multiple modes, define a separate Integration Lines for each mode since the orientation of the electric field will vary.

About Calibration Lines

- When the excitation field pattern at a Wave Port is computed, the direction of the field at $\omega t=0$ is arbitrary and can point in one of at least two ways. The Integration Lines calibrate the port by defining the preferred direction or the reference orientation. Be sure to define Integration Lines for each Wave Port so that the preferred direction is the same relative to other ports having identical or similar cross-sections. In this way, the results of laboratory measurements (in which the setup is calibrated by removing the structure and connecting two ports together) can be duplicated.

- Because the calibration lines only determine the phase of the excitation signal and the traveling wave, the system ignores them during the Ports-Only solution

Solution Type: Driven Terminal

- The Modal S-matrix solution computed by Ansoft HFSS is expressed in terms of the incident and reflected powers of the waveguide modes. This description does not lend itself to problems where several different quasi-transverse electromagnetic (TEM) modes can propagate simultaneously. For structures like coupled transmission lines or connectors, which support multiple, quasi-TEM modes of propagation, it is often desirable to compute the Terminal S-Parameters.

- To calibrate a Wave Port, that has already been defined, with Terminal Lines:
  1. From the **Project Tree**, expand **Excitations** and double click on the Wave Port to be calibrated
  2. Select the **Terminals** tab.
  3. From the table, select the **Terminal Line** column for the first terminal and choose **New Line**.
  4. Enter the position and length of the line using one of the following methods
     - Type the start and stop points of the line directly into the x, y, or z axis fields, referenced to the working coordinates. For more information on coordinates, refer to “Getting Oriented in the Drawing Space” in Chapter **, Drawing Basics and Tips.
     - Graphically pick the points in the Design Window’s graphics area. The line is displayed as a vector; the vector indicates direction. From the **Terminal Line** column, select **Swap Endpoints** to reverse the direction of the line, if necessary.
  5. Repeat steps 3 and 4 to define and apply lines to other terminals of the current Wave Port.
  6. Click the **OK** button when you are finished defining Terminal Lines
  7. Repeat steps 1-6 to apply lines to other Wave Ports.

About Terminal Lines

- The Terminal S-Parameters represent the linear combination of nodal voltages and currents for the Wave Port. From the nodal voltages and currents, the admittance, impedance, and pseudo-S-matrices can be determined.

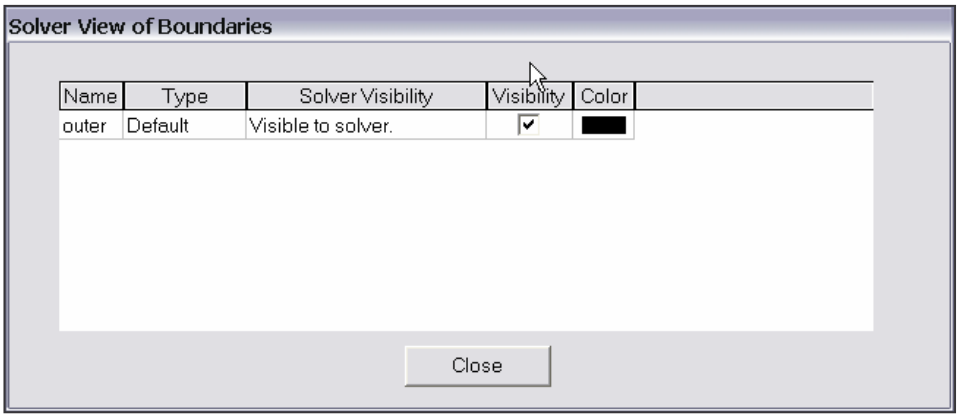
- By defining a Terminal Line for each conductor across a port, Ansoft HFSS will automatically convert the Modal Solution to its equivalent Terminal Solution.
  - In general a single terminal line is created from the reference or “ground” conductor to each port-plane conductor.

  - The polarity reference for the voltage is established by the arrow head(+) and the base(-) of the terminal line. If you decide to create terminal lines, they must be defined for every port and every terminal on the port

Considerations for Defining Wave Ports

Wave Port Locations

- It is recommended that only surfaces that are exposed to the background be defined as Wave Ports. The background is given the boundary name *outer*. Therefore a surface is exposed to the background if it touches the boundary *outer*. You can locate all regions of outer by selecting the menu item **HFSS, Boundary Display (Solver View).** From the **Solver View of Boundaries**, check the **Visibility** for *outer*.



Interior Wave Ports

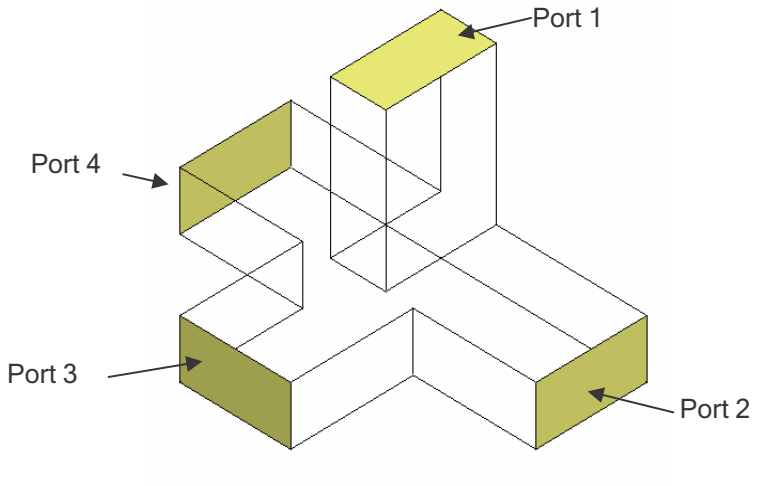
- If you want to apply Wave Ports to the interior of a structure, you must create an inner void or select the surface of an interior object that is assign a perfect conductor material property. Inner voids are automatically assigned the boundary outer. You can create an inner void by surrounding one object entirely with another object, then subtracting the interior object.

Ports are Planar

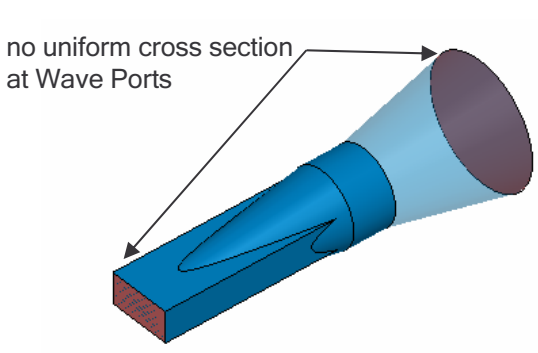
- A port must lie in a single plane. Ports that bend are not allowed. For example, if a geometric model has a curved surface exposed to the background, that curved surface cannot be defined as a port.

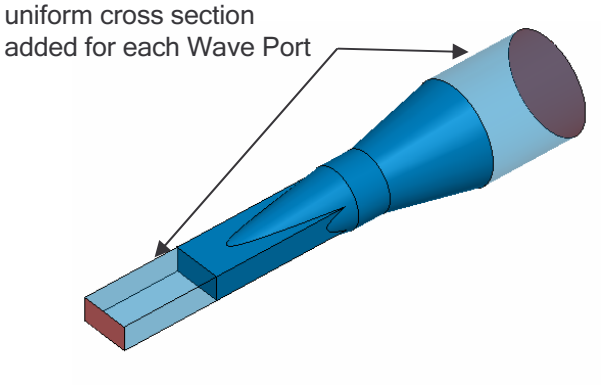
Wave Ports Require a Length of Uniform Cross Section

- Ansoft HFSS assumes that each port you define is connected to a semi-infinitely long waveguide that has the same cross section as the Wave Port. When solving for S-parameters, the simulator assumes that the structure is excited by the natural field patterns (modes) associated with these cross sections. The following figures illustrate cross sections. The first figure shows regions that have been defined as Wave Ports on the outer conductive surface of a structure.



- In the next figure, cross sections must be added to the structure. The waveguide on the left is not modeled correctly because it does not contain a length of uniform cross section at either Wave Port. To model it correctly, add a length of uniform cross section at each Wave Port, as shown on the waveguide to the right.





- The length of the uniform cross section must be long enough to allow non-propagating modes to die out. This ensures accurate simulation results. For example, if a non-propagating mode takes approximately one-eighth of a wavelength to die out, either because of losses or because it is an evanescent mode, then you should make the uniform cross section one-eighth of a wavelength. Otherwise, you must include the effects of that higher-order mode in the simulation.

- Discontinuities placed close to the port may also cause non-propagating modes to reach the port. Placing the port too close to discontinuities will cause erroneous results since the boundary condition at the port will prevent the simulated solution from matching the actual solution (i.e. The system forces the field at each port to be a linear combination of the modes you request). The energy from the non-propagating modes that reach the port will affect the apparent energy in the dominate mode and produce erroneous results.

- The modes attenuate as a function of $e^{-\alpha z}$, assuming the wave propagates in the z-direction. Therefore, the required distance (uniform port length) depends on the value of the modes propagation constant.

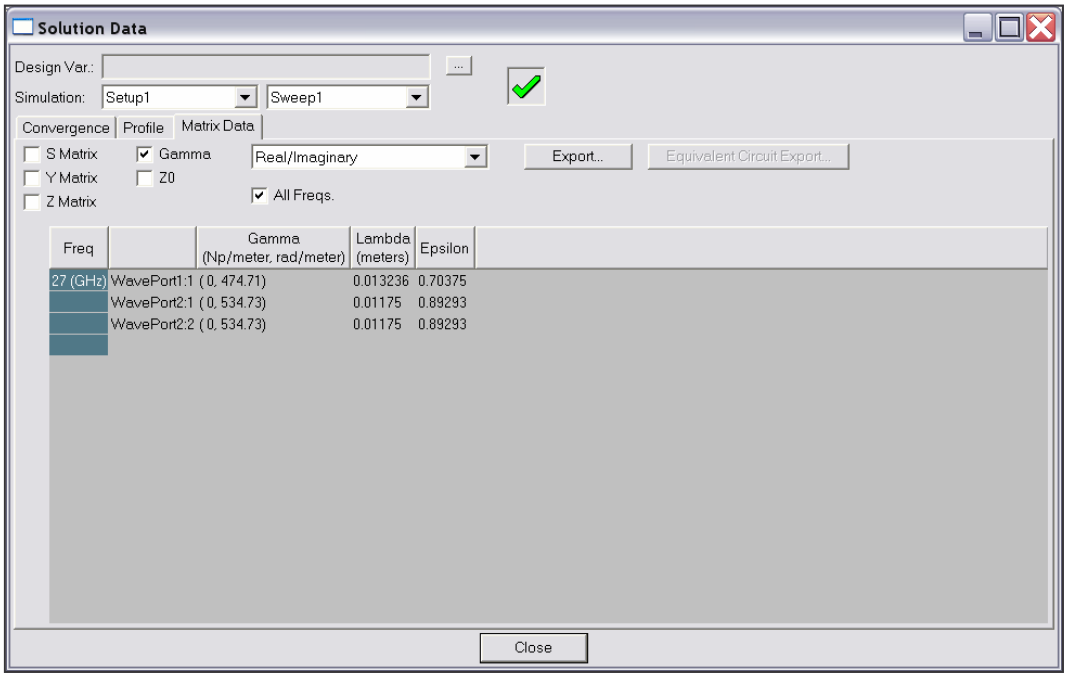
- When the Wave Ports lengths are correct, for modes that are included in the simulation there is a perfect matched condition at a Wave Port, as if the waveguide extended to infinity. For modes that are not included in a simulation, a Wave Port appears as a perfect conductor.

Wave Ports and Multiple Propagating Modes

- Each higher-order mode represents a different field pattern that can propagate down a waveguide. In general, all propagating modes should be included in a simulation. In most cases, you can accept the default of 1 mode, but where propagating higher-order modes are present you need to change this to include higher-order modes. If there are more propagating modes than the number specified, erroneous results will be generated. The number of modes can vary among ports.

Propagating Modes

- Propagating modes are those that have a propagation constant, β (rad/m), that is greater than their attenuation constant, α (Np/meter). One way to determine which modes need to be modeled is to set up the problem with multiple modes and generate a solution with no adaptive passes. Then, inspect the complex propagation constant, $\gamma = \alpha + \beta$, associated with each mode. To inspect the complex propagation constant (Gamma) after an analysis has been performed:
  1. From the **HFSS, Analysis Setup** menu, select **Matrix Data.**
  2. A dialog similar to the one shown below will appear, check **Gamma** and change the display type to **Real/Imaginary.**



- Each additional mode at a port results in an additional set of S-parameters. For example, if you are analyzing two modes at each port in a three-port structure, the final result is a 6x6 S-matrix. In general, an n-port solution is the total number of excitations of all ports, the number of modes, plus the number of sources.

- If you choose not to include some higher-order modes in a simulation, make sure the cross sections on the Wave Ports are long enough so that the modes die out and are not reflected back.

- Wave Ports and Symmetry Planes – The Impedance Multiplier
  - When a ports size is reduced because of a symmetry plane, the impedance needs to be adjusted to account for the loss of voltage and power-flow.
    - A Perfect E Symmetry plane must be adjusted by a factor of 2.
      - Such models have half the voltage differential and half the power flow of the full structure, resulting in impedances that are half of those for the full structure.
    - A Perfect H Symmetry plane must be adjusted by a factor of 0.5.
      - Such models have the same voltage differential but half the power flow of the full structure, resulting in impedances that are twice those for the full structure.
    - If the structure has a combination of Perfect E and Perfect H Symmetry planes, adjust accordingly.
      - For instance, you do not have to enter an impedance multiplier for a structure with both a Perfect E and Perfect H boundary since you would be multiplying by 0.5 and 2.

This page is intentionally left blank.

Analysis Setup

- This chapter provides details on Analysis in the Ansoft HFSS v.9.0 software such as:
  - Add Solution Setup
    - Adapt Frequency
    - Convergence Criteria
    - Initial Mesh Options
    - Adaptive Options
    - Low-Order Basis Functions
  - Setup
    - Properties
    - Add Sweep
    - Sweep – Properties and Types of Sweeps

Adaptive Meshing

The adaptive meshing constructs a mesh that conforms to the electrical performance of the device. By employing adaptive meshing, the mesh is automatically tuned to give the most accurate and efficient mesh possible. Without adaptive meshing, the responsibility of generating the correct mesh would be left to the user. This is both error prone and tedious. With Adaptive Meshing you will know the answer is right the first time.

How it Works

- The adaptive meshing algorithm searches for the largest gradients in the E-field or error and sub-divides the mesh in those regions. It also targets singularities, such as the edge of a conductor, as locations to add extra elements.
- The mesh growth for each adaptive pass is controlled by the Tetrahedron Refinement in **Setup Solution (Advanced Tab)**. You should notice, that the **Tet. Refinement** is a percentage. This ensures that between each pass the mesh is sufficiently perturbed and guarantees that you will not receive false convergences.
- After the mesh has been refined, a full solution is performed and the process is repeated until convergence

Convergence

- After each adaptive pass, HFSS compares the S-Parameters from the current mesh to the results of the previous mesh. If the answers have not changed by the user defined value or Delta S, then the solution has converged and the current or previous mesh can be used to perform a frequency sweep. If the solution has converged, then technically, the previous mesh is as good as the current mesh. In this case, Ansoft HFSS will use the previous mesh to perform frequency sweeps if they have been requested.

Delta S

- The Delta S is the default criteria used to determine mesh/solution convergence. The Delta S is defined as the maximum change in the magnitude of the S-parameters between two consecutive passes:
  - $Max_{ij}[mag(S^{N}_{ij} - S^{(N-1)}_{ij})]$, where i and j cover all matrix entries and N represents the pass number
  - Since this is the magnitude of a vector quantity, it can vary between 0 and 2

Adaptive Meshing (Continued)

- Since the adaptive meshing is based on the E-field, choosing the proper adapt frequency can be critical. Like anything in engineering, there are exceptions to every rule, but in general, the following tips will help you select the correct adaptive frequency.

Broadband Structures

- For broadband structures, the end frequency should be used since the finer mesh should be valid at all lower frequency points.

Filters

- For filters or narrow-band devices, a frequency within the pass-band or operating region should be used since in the stop-band the E-field is only present at the ports.

Fast Frequency Sweeps

- For Fast Frequency Sweeps, typically use the center of the frequency band. The Fast Frequency Sweep uses the mesh/solution at the adaptive frequency point. Since the error in the Fast Frequency Sweep typically increases as you move away from this point, the center of the frequency band is usually the preferred solution frequency to extrapolate the entire band from. It is also important to center the frequency sweep around a center point that will produce an adequate mesh. This is especially true for very high Q devices such as narrow-band filters. If the center frequency is not in the filters pass-band, the bandwidth and resonant frequency will not be accurate.

Full-Wave Spice Export

- For Full-Wave Spice problems, use the Knee Frequency ($F_{knee} \approx 0.5/rise_time$) to adapt to convergence. Then perform 2-5 more frequency points to adapt at. The additional points should be selected between the Knee Frequency and the maximum frequency (Only needs 2-3 passes per frequency point).
  - Frequencies below the Knee Frequency should have the largest impact on the time-domain, therefore the Knee Frequency is used for the primary adaptive meshing. Unfortunately, the mesh needed at the higher frequencies may not be resolved enough without performing the additional adaptive mesh passes.
  - Due to the large bandwidths, typically you will use an Interpolating Sweep. Using multiple frequency sweeps and combining the results may also be useful.

- Adaptive Meshing (Continued)
  - High-Speed Digital/Signal Integrity
    - For high-speed digital simulations you are interested in the performance over a broad frequency range. To determine the frequency bandwidth that you need to simulate over, the following guide is recommended:
      - BW ≥ $5*F_{knee}$, where $F_{knee} \approx 0.5/rise_time$
    - In general, all of the frequencies below the F_{knee} frequency have the largest impact on the digital signal. Therefore a good high-speed digital design will have a clean, well matched response up to at least F_{knee}.
  - High-Speed Digital (SPICE Export)
    - To export to SPICE, for transient simulations, a large bandwidth is required ($\geq 5*F_{knee}$). The reason for this is that the F_{knee} is an approximation, plus you want more then 1 sample point over the rise-time.
    - When exporting to SPICE, the low frequency is also important. You want to get as close to DC as possible. Every port setup and every problem is a little different with regards to how low you can solve in Ansoft HFSS. 99% of the time you can simulate to at least 100MHz. Below that is trial an error. When you export to Full-Wave SPICE, the DC component will be extrapolated from the lowest frequency in the solve. Therefore, going from 1GHz to DC is not going to give a very accurate extrapolation.
    - Frequency spacing. The setup for Full-Wave Spice recommends that the minimum frequency be used as the spacing. This usually results in 1000-3000 points depending on the bandwidth of the sweep.
  - High-Speed Digital (Adaptive Meshing)
    - Since the simulation bandwidths can be so large, determining the proper adaptive mesh frequency can be very difficult. The following technique is recommended:
      1. Adapt at the F_{knee} until convergence (Delta S 0.02 to 0.01)
      2. Pick 2-3 frequency points above F_{knee} to adapt at
         - Don't run these to convergence, just do 3-5 passes
      3. Solve the Frequency Sweep
         - For large bandwidths, either break up the frequency sweep and/or use Interpolating Sweeps

Adaptive Meshing (Continued)

High-Speed Digital (Why this technique?)

- If the frequency points below Fknee have the biggest impact on the digital response, then meshing at that frequency should produce an accurate mesh for all frequency points below it.
- If you simulate most passive devices over a large enough bandwidth, they start to exhibit a low pass filter response. So the higher frequency content contributes less to the time-domain simulation in addition to being filtered out by the devices frequency response. This is why we don't worry as much about the higher frequency adaptive passes converging. To appreciate the significance of this you also have to take into account how HFSS does its adaptive meshing. It is done by finding the highest gradient of the electric field. So if the device behaves as a filter and we are adapting outside the pass band of the filter, we are focusing most of the mesh only at the ports.
- Now if your device performs well beyond Fknee, there is no harm in adapting at a higher frequency until convergence. Unfortunately, for real problems(10-40Gb/s) the design challenge is to get the device to work at least up to the knee frequency.
- For these large Bandwidth sweeps, use the Interpolating Sweep. This sweep is based on the discrete sweep, but it adaptively picks discrete points and curve fits. So it takes less discrete points to represent a large bandwidth. The interpolating sweep can use either a polynomial fit or a rational function. So it can help to break up the sweep. Up to the knee frequency works good for a polynomial fit and above that the rational function will work better. The interpolation sweep will always pass through the start/end frequency points so if you don't change the mesh and make sure the start and end frequency points match up for your sweeps, they combine together without a problem.

Adaptive Meshing (Continued)

High-Speed Digital (Design Flow)

- Up until this point, the entire discussion of High Speed Digital was based on the assumption that you are exporting every simulation to SPICE. There are many stages in the design cycle and going to SPICE is not required for every part of the design. If you have resonance's, mismatch, coupling, etc below the Fknee, these should be addressed in the field solver prior to running SPICE simulations. Therefore, the sweep only needs to be done to 1.5-2x Fknee to investigate the performance. In addition, for engineering design purposes, you may not need to go below 1GHz.
- During these early stages of design, you may also want to use the Fast Frequency Sweep since you can get S-Parameters and fields for every frequency in the Sweep. This allows you to visualize any resonance or coupling you may be seeing in the S-Parameters.
- Another useful tool for package/board analysis is the EigenMode solver. In many instances, resonances caused by the power and ground plane nets are the largest contributor to designs problems. By removing everything except the power/ground nets, the Eigenmode solver can be used to quickly identify resonances.

Setting Convergence

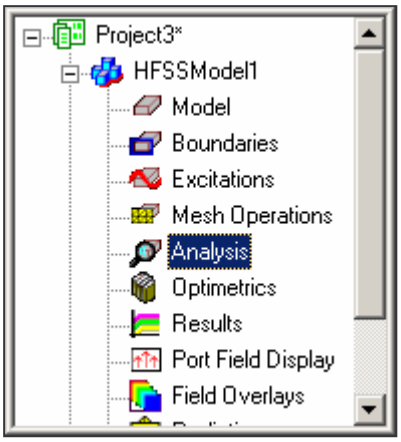
- It is very common to request too much accuracy when defining the Delta S. Remember! The manufacturing process, the laboratory equipment, and the measurement process all have inherent errors associated with them. Asking HFSS to provide a level of accuracy that is orders of magnitudes greater then what you can obtain in the real-world does not buy you anything other then extra simulation time. Instead, use the Convergence Monitoring and good engineering judgment to determine when to terminate the adaptive meshing process and how to set the Delta S. In general a Delta S of 0.02(2%), which is the default, or as low as 0.01(1%), is sufficient.

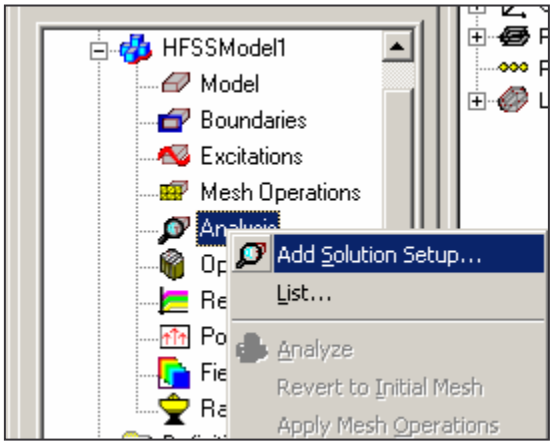
Solution Convergence – S-Matrix vs. Fields

- The convergence criteria is based on the S-Matrix. Typically, the S-Matrix converges prior to field quantities. That is to say if you are looking for the absolute field value within the problem region, chances are you will need to solve several more adaptive passes in order to see the same convergence that was obtained for the S-Matrix. This will also depend on the field quantity you are solving for. Ansoft HFSS solves for the E-field directly. From the E-field it calculates the H-field and from the H-field it calculates current. Therefore, the field quantities will also converge with varying mesh densities.

Adding a Solution Setup

- In order to perform an Analysis in Ansoft HFSS a Solution Setup must be added. To do this, right click on Analysis in the Ansoft HFSS model tree.



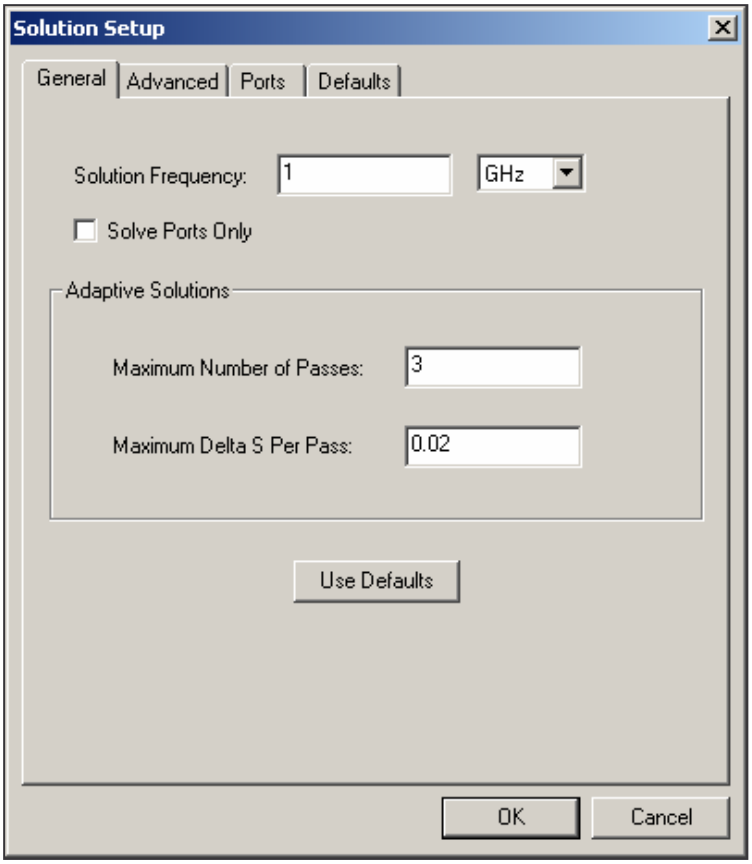


- By default, the General Tab will be displayed. The Solution Frequency and the Convergence Criteria are set here.

Adding a Solution Setup (Continued)

General Tab

- **Solution Frequency** – This frequency point is used by the adaptive mesher to automatically refine the mesh to electrical performance of the device.
  - **Solve Ports Only** - The Port Solution uses an arbitrary, adaptive 2D eigenmode solver to determine the natural frequencies or modes that will be used to excite the structure. The ports only solution can be used to calculate only the modal field patterns for the 2D cross sections defined to be ports. This is useful for determining the number of modes, modal fields, the port length, and/or proper port setup prior to running a full solution.
- **Adaptive Solutions**
  - **Maximum Number of Passes** - This number controls the maximum number of passes the adaptive mesh routine will perform as it attempts to satisfy the convergence criteria.
  - **Maximum Delta S Per Pass** – This number defines the convergence criteria for the adaptive meshing process.

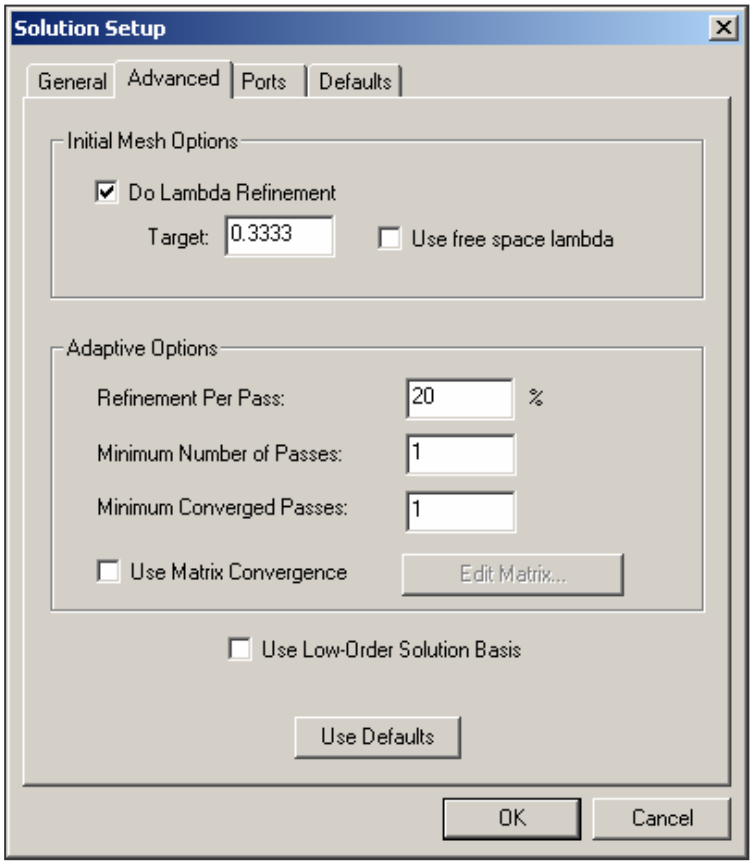


Adding a Solution Setup (Continued)

Advanced Tab

Initial Mesh Options

- **Lambda Refinement** - The Initial Mesh is based only on the 3D solid model, it has no bearing on the electrical performance of the device to be simulated. The Lambda Refinement process refines the Initial Mesh until most mesh element lengths are approximately one-quarter wavelength for air and one-third wavelength for dielectrics. A wavelength is based on the Single Frequency value entered in the **Solution Frequency**. In almost all cases Lambda Refinement should be used.
- **Use Free Space Lambda** – This will force the lambda refinement to target a mesh size approximately one-quarter of a wavelength for air. The material properties of objects will be ignored. This may be useful in applications that have dielectrics with very high conductivities. Brain tissue or salt water are examples of materials that will produce very high mesh counts even though the RF penetration into the material will be limited to a region very close to the surface.

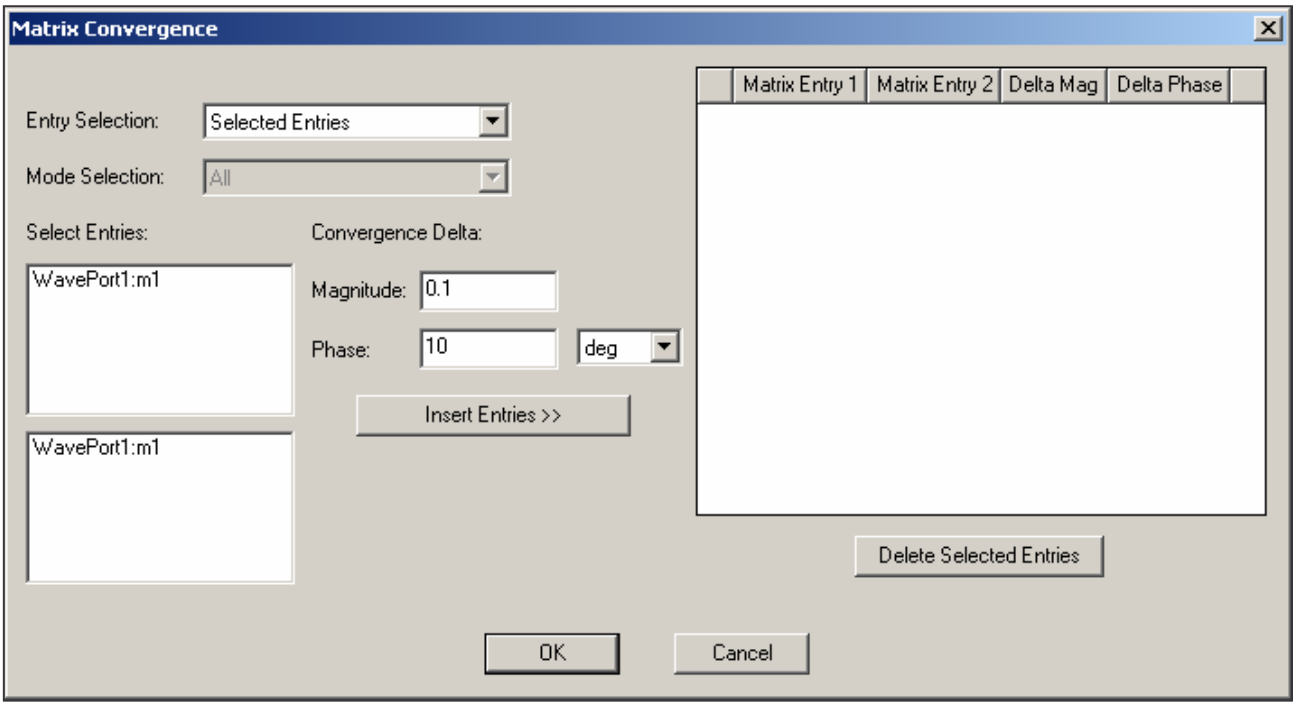


Adding a Solution Setup (Continued)

Advanced Tab (Continued)

Adaptive Options

- **Refinement Per Pass** -The mesh growth for each adaptive pass is controlled by the Refinement Per Pass. The Refinement Per Pass is a percentage. This ensures that between each pass the mesh is sufficiently perturbed and guarantees that you will not receive false convergences.
- **Minimum Number of Passes** - An adaptive analysis will not stop unless the minimum number of passes you specify has been completed, even if convergence criteria have been met
- **Minimum Converged Passes** - An adaptive analysis will not stop unless the minimum number of converged passes you specify has been completed. The convergence criteria must be met for at least this number of passes before the adaptive analysis will stop
- **Use Matrix Convergence -** You can specify different stopping criteria for specific entries in the S-matrix. This is done by checking the **Use Matrix Convergence** box. The adaptive analysis will continue until the magnitude and phase of the entries change by an amount less than the specified criteria from one pass to the next, or until the number of requested passes is completed.



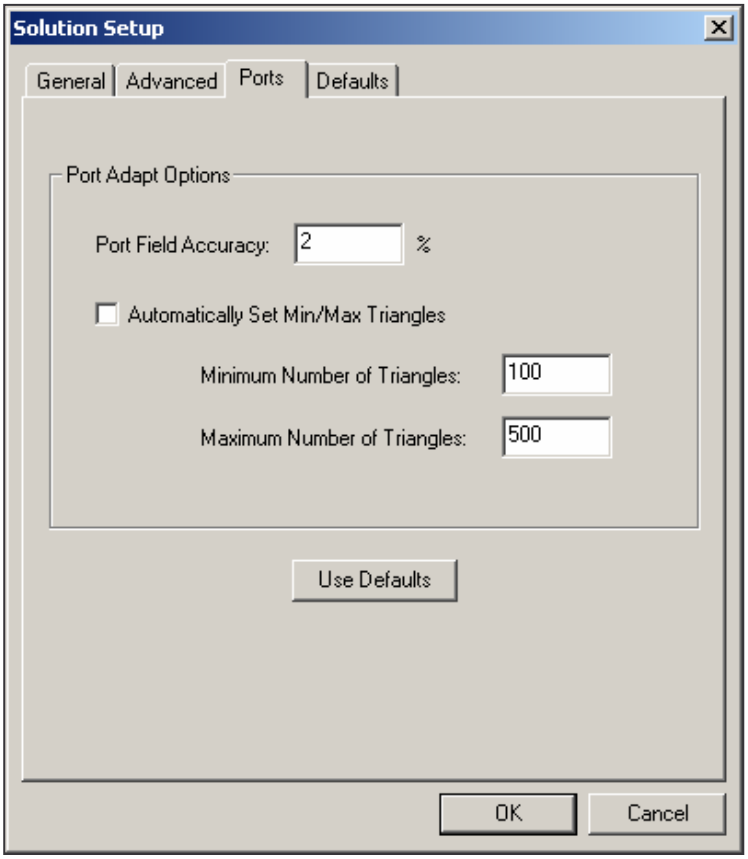
- Adding a Solution Setup (Continued)
  - Advanced Tab (Continued)
    - **Use Low-Order Solution Basis** – The Low Order Basis Functions reduce the default second-order basis function to a linear basis function. It is intended for simulations in which the edge to edge spacing between tetrahedron is electrically small. In this situation, the basis function, and hence the number of unknowns, can be reduced. For this assumption to be valid, the edge lengths for all tetrahedron in the model should be on the order of 1/20th of a wavelength. In versions prior to v9.0, this was set with a system environment variable called ZERO_ORDER.
      - Common Applications :
        - On-Chip Spiral Inductors, Capacitors, Transformers, etc.
        - Package Analysis – Flip-Chip, BGA, etc.

Adding a Solution Setup (Continued)

Ports Tab

Port Adapt Options

- Port Field Accuracy - Usually, the default **Port Field Accuracy** value is adequate. You may want improved port accuracy under the following conditions:
  - You are interested primarily in the port impedances. Port impedances are computed as part of the port solution.
  - You need to lower the noise floor to catch S-parameters that are expected to be in the -70 dB range.
- Refining the mesh at the ports causes HFSS to refine the mesh for the entire structure as well. This occurs because it uses the port field solutions as boundary conditions when computing the full 3D solution. Therefore, specifying too small a port field accuracy can result in an unnecessarily complex finite element mesh



Adding a Solution Setup (Continued)

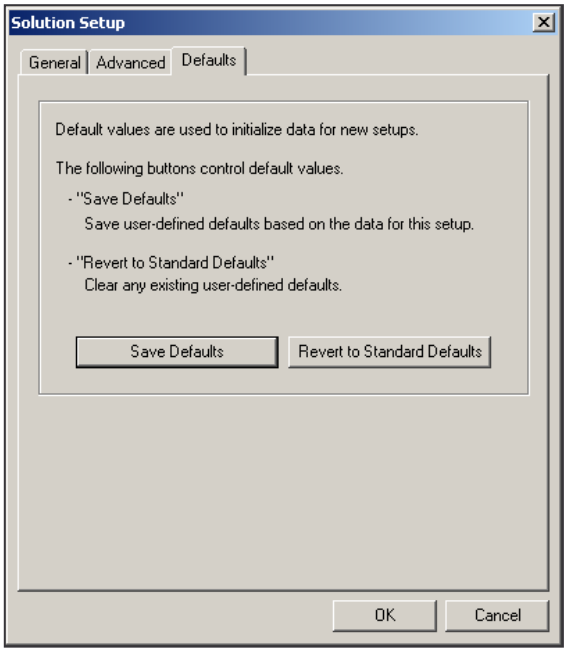
Ports Tab (Continued)

Set Min/Max Triangles

- The mesh for each model port will be adaptively refined until it includes the minimum number of triangles. Refinement will then continue until the port field accuracy or the maximum number of triangles is reached.
  - To specify the minimum and maximum number of triangles in the port mesh, uncheck the **Automatically Set Min/Max Triangles** option.
  - Enter a value for the **Minimum Number of Triangles**. The default value is 25 for models with lumped gap ports and 90 for models with wave ports.
  - Enter a value for the **Maximum Number of Triangles**. The default value is 400.
- If you leave **Automatically Set Min/Max Triangles** selected, HFSS will determine the reasonable values for the minimum and maximum number of triangles based on the port's setup.

Defaults Tab

- The Defaults tab allows you to save the current settings as the defaults for future solution setups or revert the current settings to the standard setting.



Frequency Sweeps

- Using the converged mesh or initial mesh if no adaptive passes were requested, the swept frequency response of the device can be obtained. Ansoft HFSS provides several methods for calculating the frequency response:
  - **Discrete** – performs a full solution at every frequency using the current mesh. The time required is the single frequency solve times the number of frequency points. Fields can be displayed at any frequency within the sweep range if the Save Fields Box is checked.
  - **Fast** – uses an Adaptive Lanczos-Pade Sweep(ALPS) based solver to extrapolate an entire bandwidth of solution information from the center frequency. Very good for high-Q devices but it can not be used to solve for devices that pass through cut-off. Once the band has been extrapolated, a high number of frequency points can be calculated without a penalty. In addition, the Fields can be displayed at any frequency within the sweep range. The time and memory required to solve a fast frequency sweep may be much larger then the single frequency solve.
  - **Interpolating** – performs solves at discrete frequency points that are fit by interpolating. Ansoft HFSS determines the frequency points to solve at based on the error in the interpolation between consecutive passes. The interpolation error and maximum number of points is defined by the user in the **Edit Sweep.** As with the fast frequency sweep, the Interpolating Sweep can generate a larger number of frequency points. But you only have the field solution for the last solved frequency. The maximum solution time is the single frequency solve times the maximum number of points.

Frequency Sweeps (Continued)

Adding Points to a Fast or Interpolating Sweep

- After the Fast or Interpolating sweeps have completed, additional frequency points can be added by changing the number of points in **Edit Sweep** Clicking on **Analyze** will not resolve the entire frequency sweep, it will just calculate the additional frequency points.

Adding Solutions to Interpolating Sweeps

- If the interpolating sweep does not converge with the requested passes or you wish to change the target convergence, the values can be changed and resolved. The interpolating sweep will recalculate the two end frequencies but after that it will use the previously calculated points and continue trying to reach the target convergence.

Ports Only and Frequency Sweeps

- A discrete or interpolating frequency sweep can be used with the Ports Only solution.

Fast Frequency Sweep Ports Solve

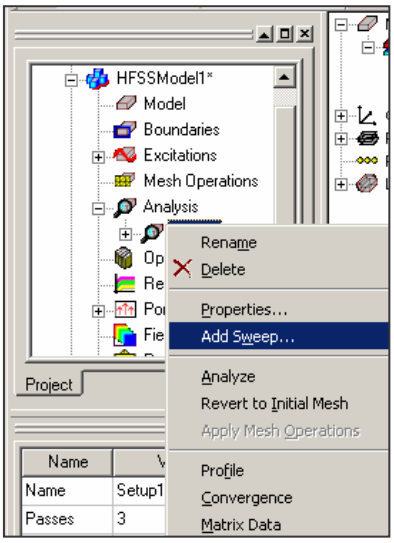
- The Fast Frequency sweep can not be used on ports that pass through cut-off. You may also experience problems if the sweep frequency approaches cut-off.

Combining Multiple Frequency Sweeps

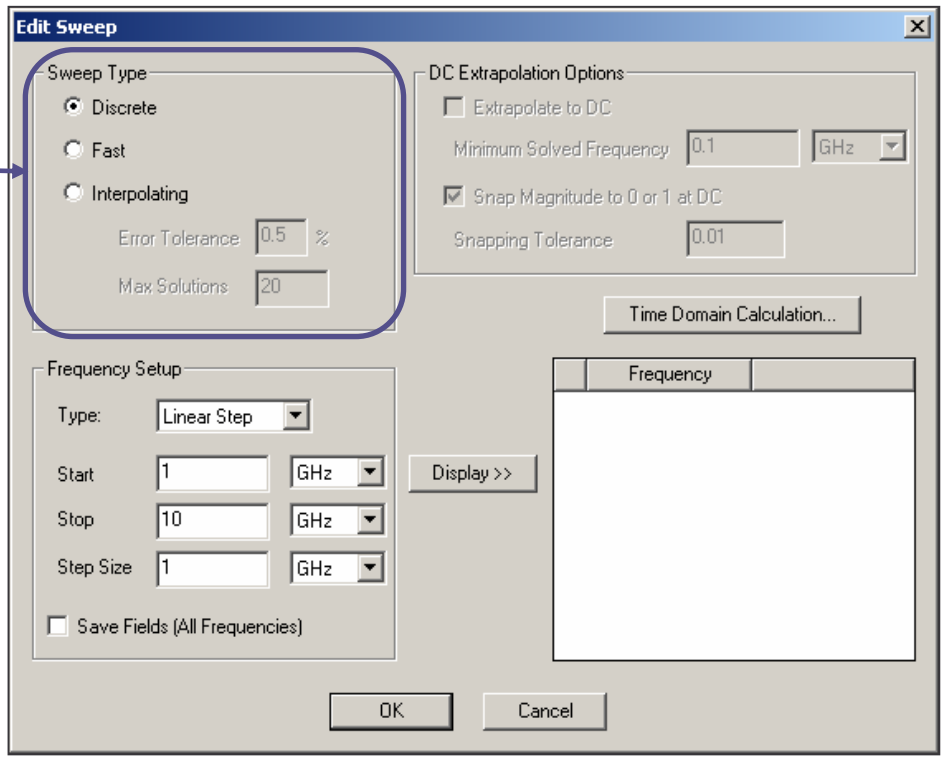
- For very large bandwidths, breaking the band into smaller frequency sweeps can improve the results. Since the Fast Frequency Sweep is extrapolated from the center frequency different error curves at the end frequencies will prevent the sweeps from aligning. The interpolating sweep will not have this problem since the solution always passes through the end frequencies (Assuming the same mesh is used).

Add Sweep

- After a Solution Setup has been added you can also add a Frequency Sweep. To do this, right-click on Setup in the HFSS Model Tree. The Edit Sweep window will appear



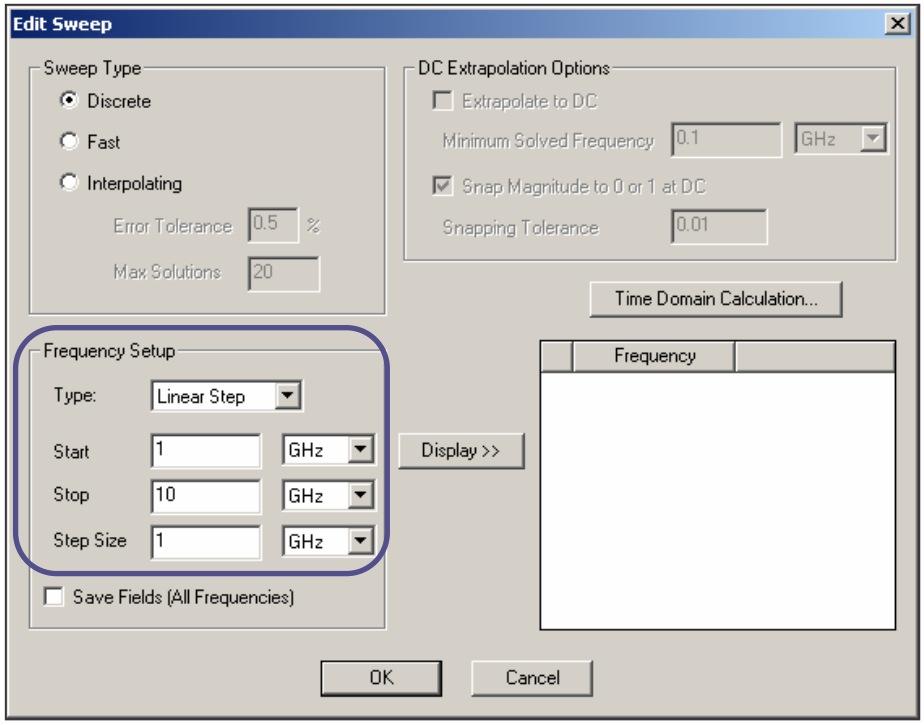
Sweep Type



Add Sweep (Continued)

Frequency Setup

- After the sweep type has been chosen, the frequencies of interest must be specified.



- There are three Frequency Setup Options:
  - Linear Step -- specify a linear range of values with a constant step size
  - Linear Count -- specify a linear range of values and the number, or count, of points within the variable range
  - Single Points -- specify a single values for the sweep definition

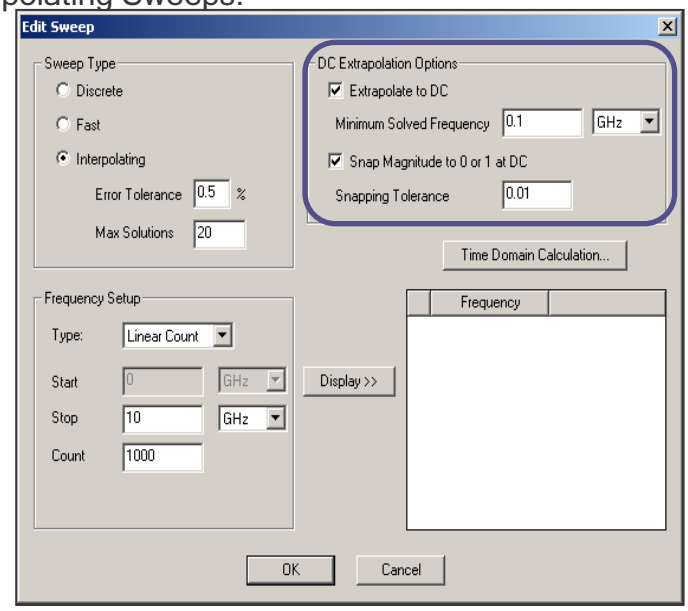
Saving Fields

- It is possible to save the Field data for every point in the Fast Sweep and the Discrete Sweep. To save the Field information make sure that the Save Fields (All Frequencies) box is checked.
- For the Interpolation Sweep, only the Field data for the last solved frequency will be available for post-processing.

Add Sweep (Continued)

DC Extrapolation Options

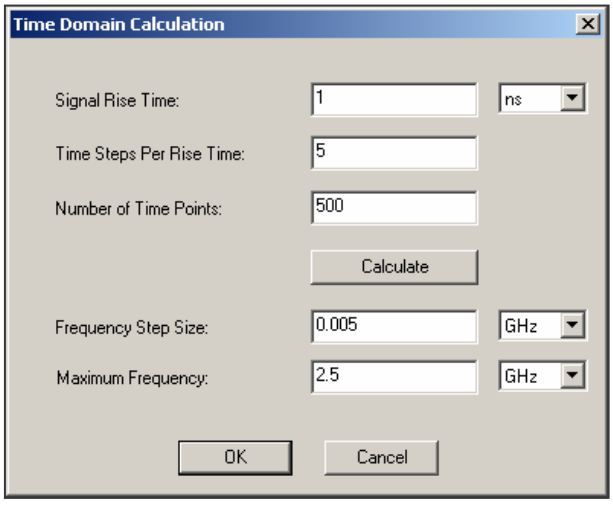
- When exporting Spice subcircuits it is necessary to include the DC point. Since Ansoft HFSS does not solve down to DC we can use DC Extrapolation. The DC Extrapolation option is available in the Discrete and Interpolating Sweeps.



Time Domain Calculation

- Ansoft HFSS can calculate the maximum frequency required to obtain an accurate time domain result. HFSS uses the following equation:

Max. Freq. = (0.5/Signal Rise Time) x Time Steps Per Rise Time



This page is intentionally left blank.

Ansoft HFSS Data Reporting

Overview

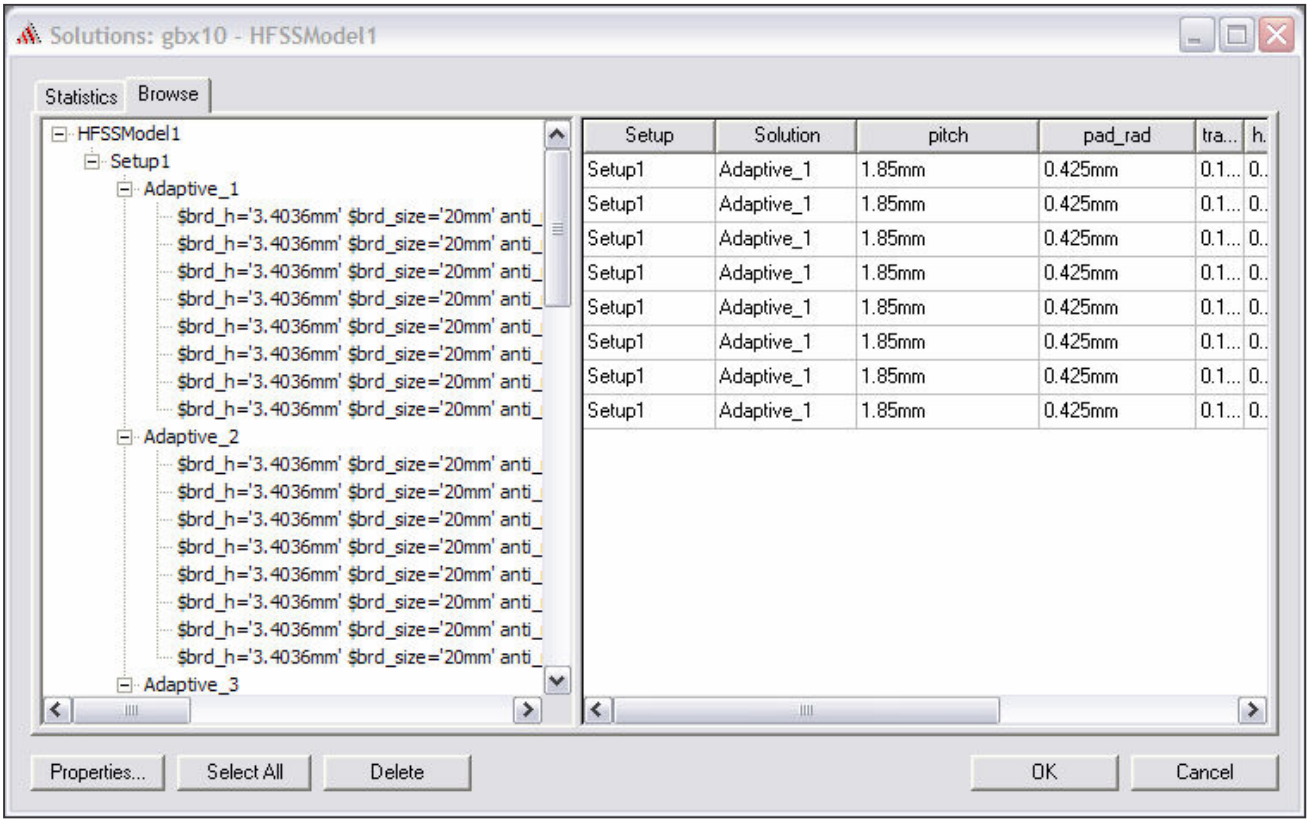
- Ansoft HFSS has very powerful and flexible data management and plotting capabilities. Once understood, it will make the whole solution process much easier, and will help craft the entire problem setup.

Topics of Discussion

- Data management
- 2D Plotting
- 3D Plotting
- Antenna characteristics
- Field Plotting

Data Management

- With every model variation that is solved, a new data entry is stored within the project results directory. This capability has implications in that the user can perform a parametric sweep of a model without needing an Optimetrics license.
  - NOTE: Automatic parametric sweeps along with the other Optimetrics functions, Optimization, Sensitivity, and Statistical analyses, DO require an Optimetrics license.
- The dataset validity can be checked by selecting ***HFSS > Results > Browse Solutions***

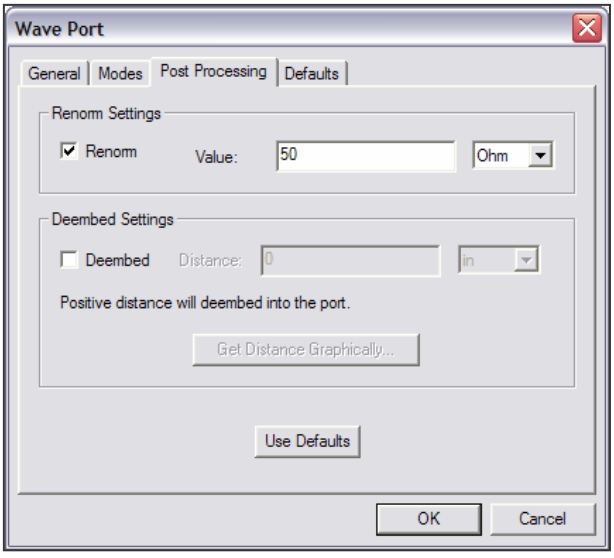


- By examining this dialog, the user can determine which parametric instances have been solved, and how many adaptive passes were necessary.

Data Management

- Post-processing steps can also be performed in the new interface for HFSS. The common operations that were previously performed in the Matrix Data module of previous releases are:
  - **Port Impedance Renormalization**
  - **Port De-embedding**
  - **Differential pair analyses**
- Other post-processing operations that used to require the Matrix Data module that are now computed by default in HFSS version 9.0 are the **Y- and Z-matrices**. Also, given that the **Solution Type** defined early on in the process, negate the necessity of computing Terminal S-parameters from Modal S-parameters.

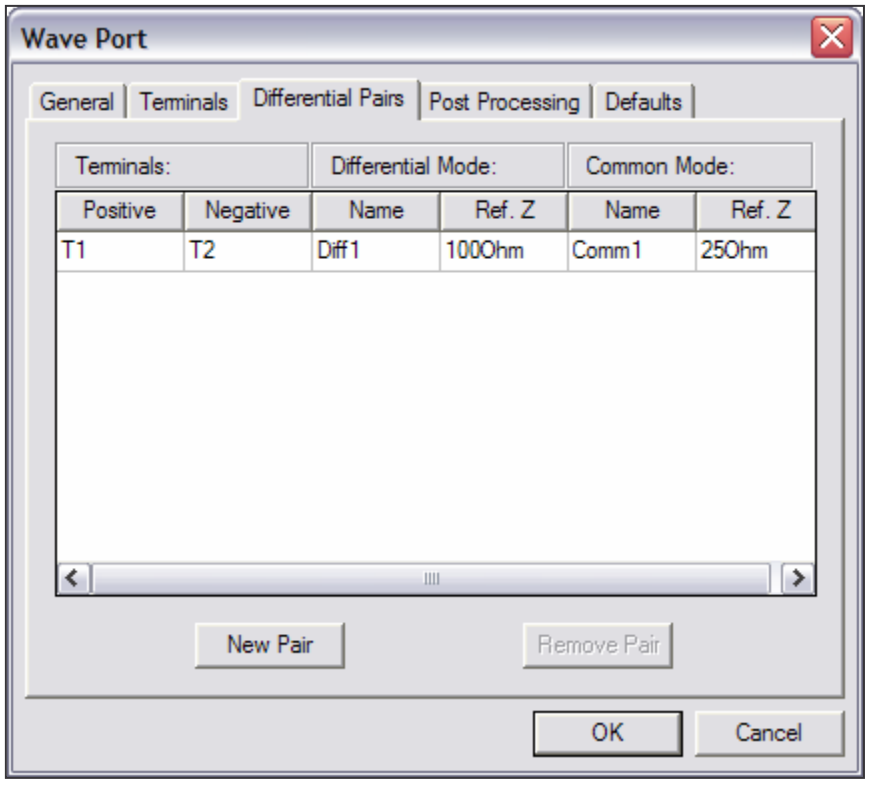
- **Port Impedance Renormalization**
  - Within the new interface, many calculations are made automatically without user intervention. The wave port renormalization impedances are specified with the port wizard. By simply editing the properties of a port once a solution is completed, the port can be re-normalized.



- **Port De-embedding**
  - This same dialog can be used to change the distance that a port will be de-embedded. The user can go back and edit this value as many times as necessary. Each time the OK button is pressed, the data, and also the plots if they exist, will be updated with the newly de-embedded data.
  - Positive values of de-embedding will move the reference plane *into* the model

Data Management

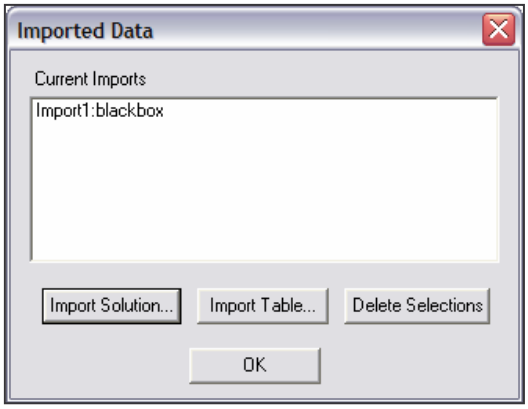
- **Differential pair analyses**
  - For multiple terminals per port, differential pairs can be created to look at differential S-parameters instead of single-ended S-parameters. This can be useful for investigating the possible conversion between differential and common mode within a given structure.
  - Within the wave port dialog, the Differential Pairs tab controls the creation of the pairs from the individual Terminals
    1. To create a differential pair, select **New Pair**, and select the terminal lines that represent the positive and negative sides
    2. From here, you can also change the **Differential** and **Common** mode impedance setting for each pair.



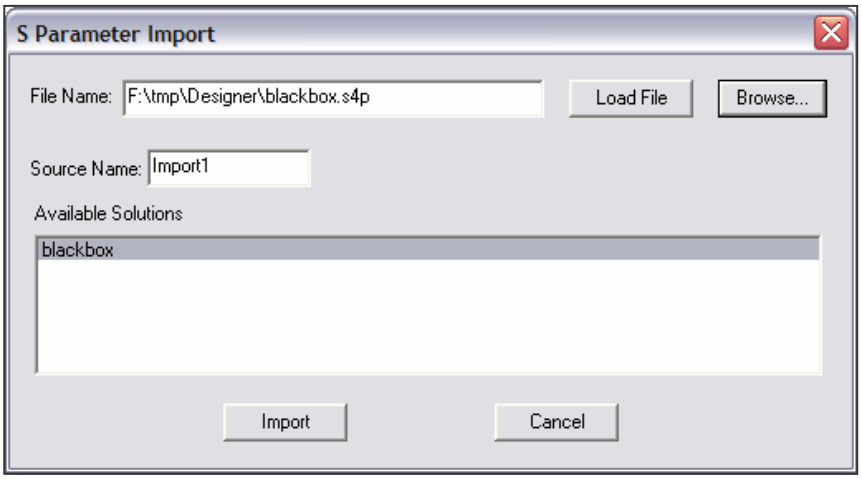
Data Management

Importing Data from other solutions

- Often it is desirable to compare the results of one simulation to the results of another simulation, HFSS, circuit, or planar, or even measurements. This can be accomplished readily within the HFSS version 9.0 desktop environment.
- **To import solutions into HFSS Desktop:**
  - NOTE: The minimum requirements for importing a solution are an existing model with at least one port.
  1. Select ***HFSS > Results > Import Solutions***, and the following dialog will appear:



  - The import options are **Import Solutions** and **Import Table**. Solutions represent S-parameter matrices in standard forms such as Touchstone and Ansoft legacy data in .szg file format. Tables are simply files containing rows and columns of data.
  2. Selecting **Import Solution** will bring up the following dialog:

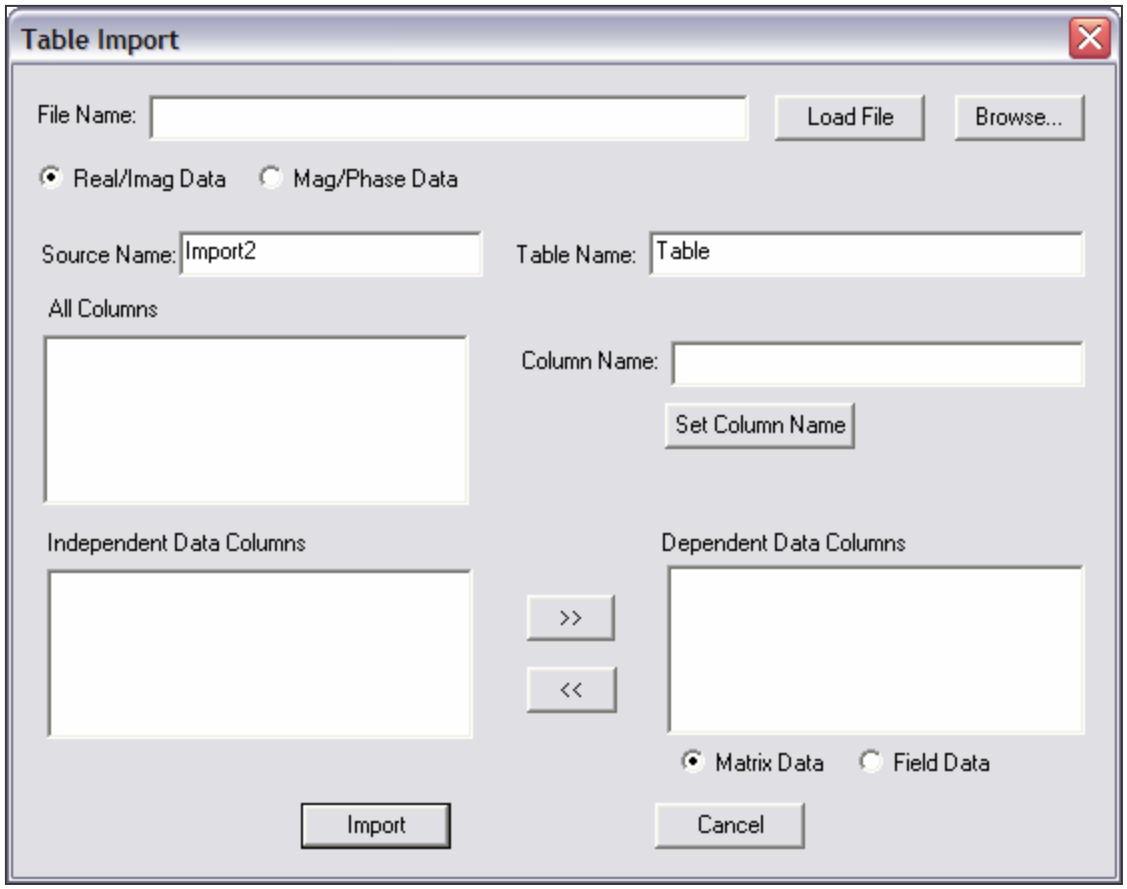


Data Management

3. From this window, select **Browse** to find the file to be imported. The acceptable file formats are: .sNp, .szg, .flp, .yNp, .zNp, and .tou.
4. Click **Open** to load the file.
5. Click **Import** to bring the data into the HFSS desktop
6. Click **OK** when you are done importing solutions

- All solutions will appear in the report editor under the **Sweep** category (more on this later)

- For Import Table, the following dialog will appear:

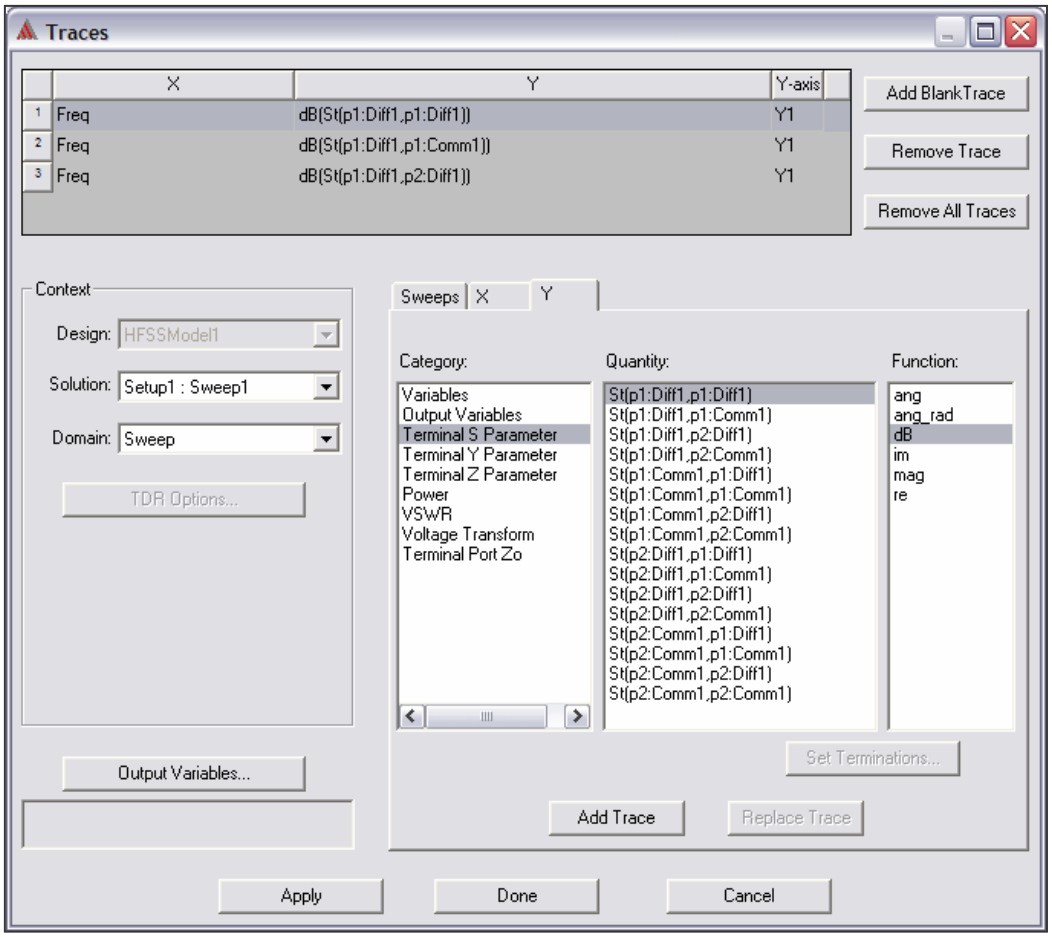


Plotting Data

- Data plotting can take a variety of forms. The most often used format is 2D Cartesian plotting, but we also have the capability to plot in 3D as well. Below is a list of all the quantities that can be plotted on various graphs. For definitions of each of these quantities, see the online help.
  - **Eigenmode solution**
    - Eigenmode Parameters (modes)
  - **Driven Modal Solution**
    - S-parameters
    - Y-parameters
    - Z-parameters
    - VSWR
    - Gamma (complex propagation constant)
    - Port Zo
  - **Driven Terminal Solution**
    - S-parameters
    - Y-parameters
    - Z-parameters
    - VSWR
    - Power (at port)
    - Voltage Transform matrix (T)
    - Terminal Port Zo
  - **Fields**
    - Mag_E
    - Mag_H
    - Mag_Jvol
    - Mag_Jsurf
    - ComplexMag_E
    - ComplexMag_H
    - ComplexMag_Jvol
    - ComplexMag_Jsurf
    - Local_SAR (Specific Absorption Rate)
    - Average_SAR
  - **NOTE**: For all Field plots, a polyline or surface must be selected before creating the Field plot.

Plotting Data

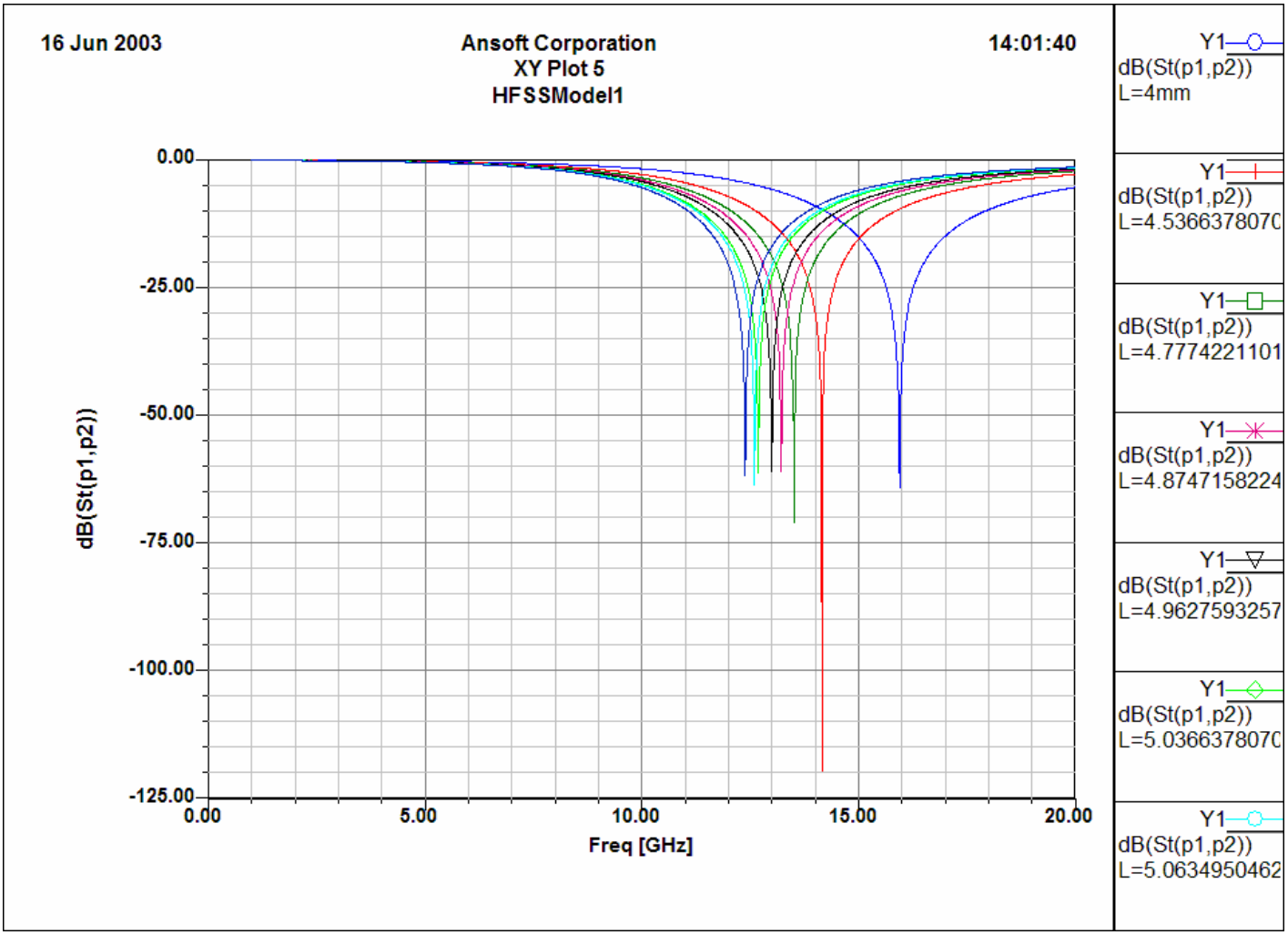
- **Types of Plots:**
  - Rectangular Plot
  - Polar Plot
  - 3D Rectangular Plot
  - 3D Polar Plot
  - Smith Chart
  - Data Table
  - Radiation Pattern
- **To Create a Plot:**
  1. Select HFSS > Results > Create Report
  2. Select Report Type and Display Type from the selections above
  3. Click OK and the Report Editor will be displayed – we will go over the options in this dialog on the next page.



Plotting Data

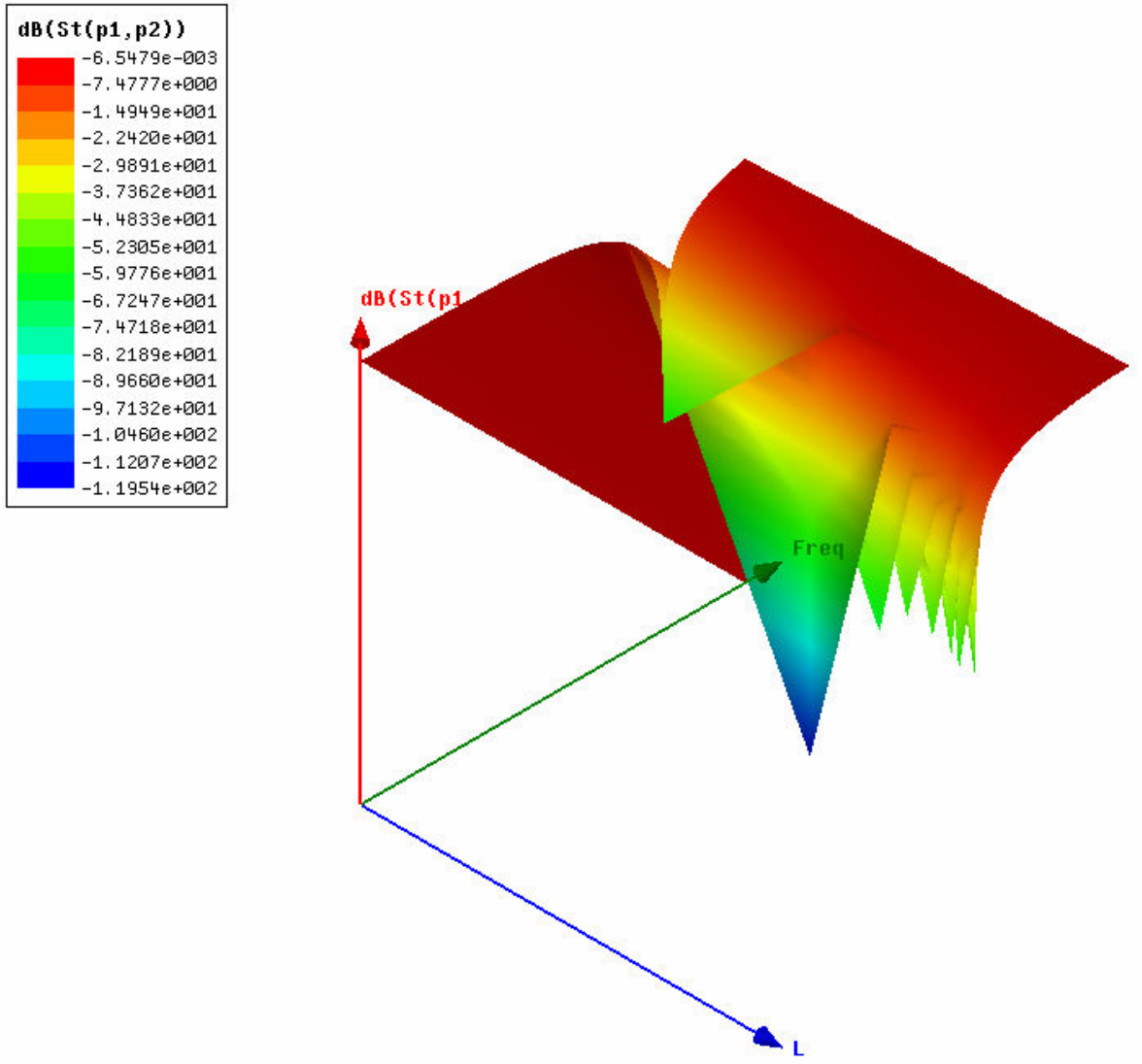
- **Creating a Plot** (cont'd)
  4. **Context**
     - **Design** – choose from available designs within a project
     - **Sweep** – choose from available sweeps including adaptive passes and imported data
     - **Domain** – defaults to Sweep, but can be switched to Time domain for plotting S-parameters with an impulse or step response.
  5. **Sweep / X / Y Tabs**
     - **Sweep** – controls the source of the independent variable in the plot.
       - **NOTE:** By default, the Report editor selects **Use Current Design and Project variable values**. This will select the primary sweep of frequency usually, and the current simulated values of the project variables
       - To display a plot with multiple traces for different variable values, change this setting to **Sweep Design and Project variable values** You can then change the primary sweep from frequency to a variable if desired. This is useful for looking at S21 versus stub length, for example. Simply select the value in the **Name** column next to **primary sweep**, and change the value to whatever you desire.
         - You can also uncheck the **All Points** block to select which values of a variable are to be plotted.
     - **X** – controls any functional operator on the independent variable
     - **Y** – select the value to be plotted and any operator
  6. Select **Add Trace** for as many values as you would like to plot
  7. Select **Done** when finished
- An example of a multi-trace plot of the sweep tab shown on the previous page is shown next

Plotting Data



Plotting Data

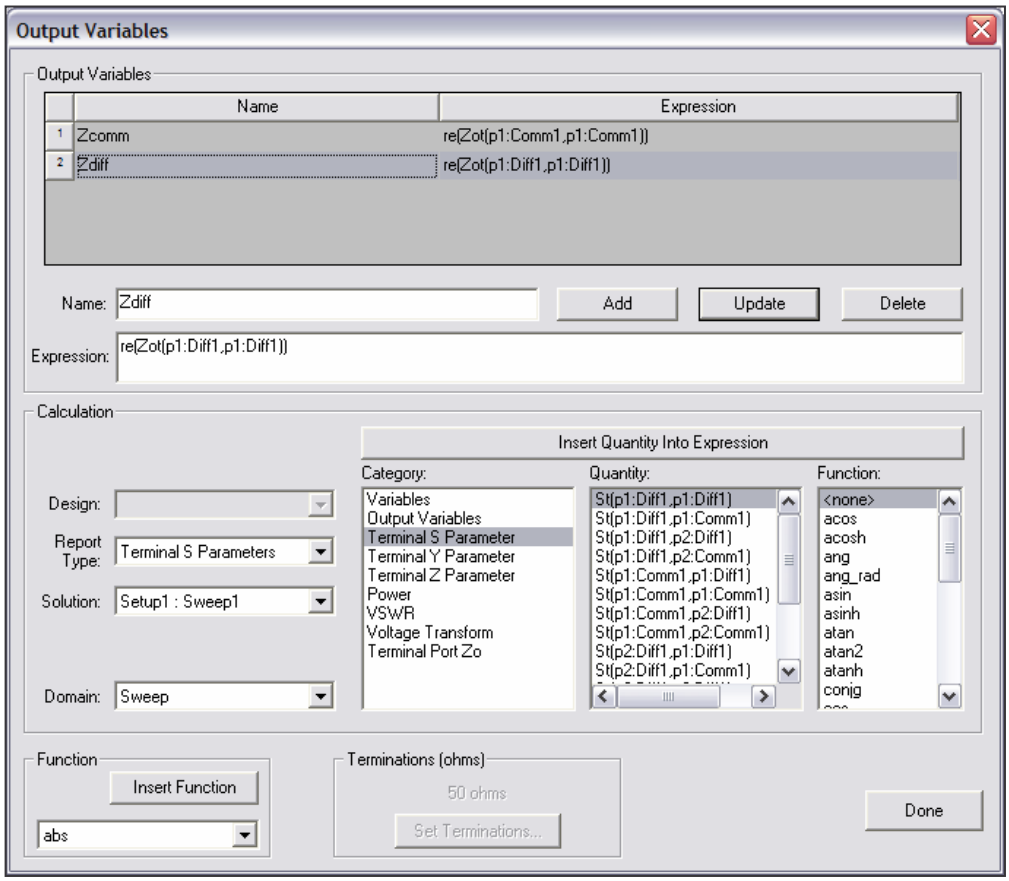
- 3D Plots – simply add a third dimension such that instead of plotting a family of curves, you can plot a 3D surface that represents your data changing with two independent variables. Below is a 3D plot of the previous family of curves.



Plotting Data

Output Variables

- In addition to being able to plot the built-in solved quantities, you can also create your own by using output variables.
- Clicking on the Output Variables button in the Traces dialog shown before, brings up the following dialog:

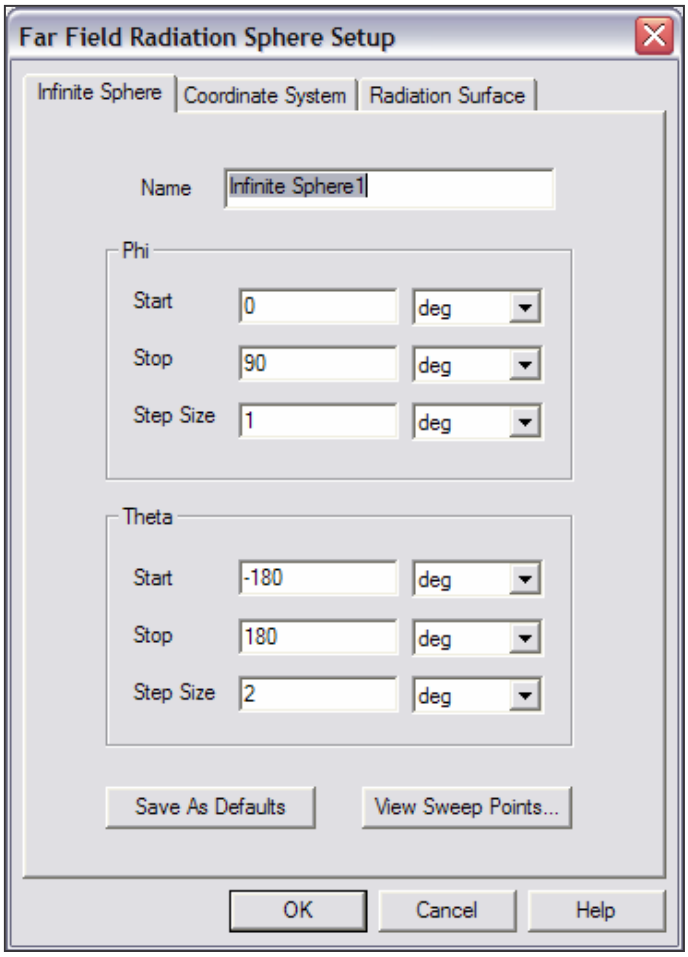


- **To create an output variable:**
  - Type a name into the **Name** box
  - Create an equation in the **Expression** box
  - Click **Add**
- In the example show above, we have created an output variable that makes an approximation for Inductance of a spiral inductor. We could also have created an equation to calculate Q as well.
- **NOTE**: These output variables can also be used for optimization.

Data Plotting – Special Case – Antenna Parameters

- Since antenna parameters require a special computation setup to determine the region over which the fields are to be computed, displaying antenna parameters is a two step process.
- Create an Infinite Sphere setup:

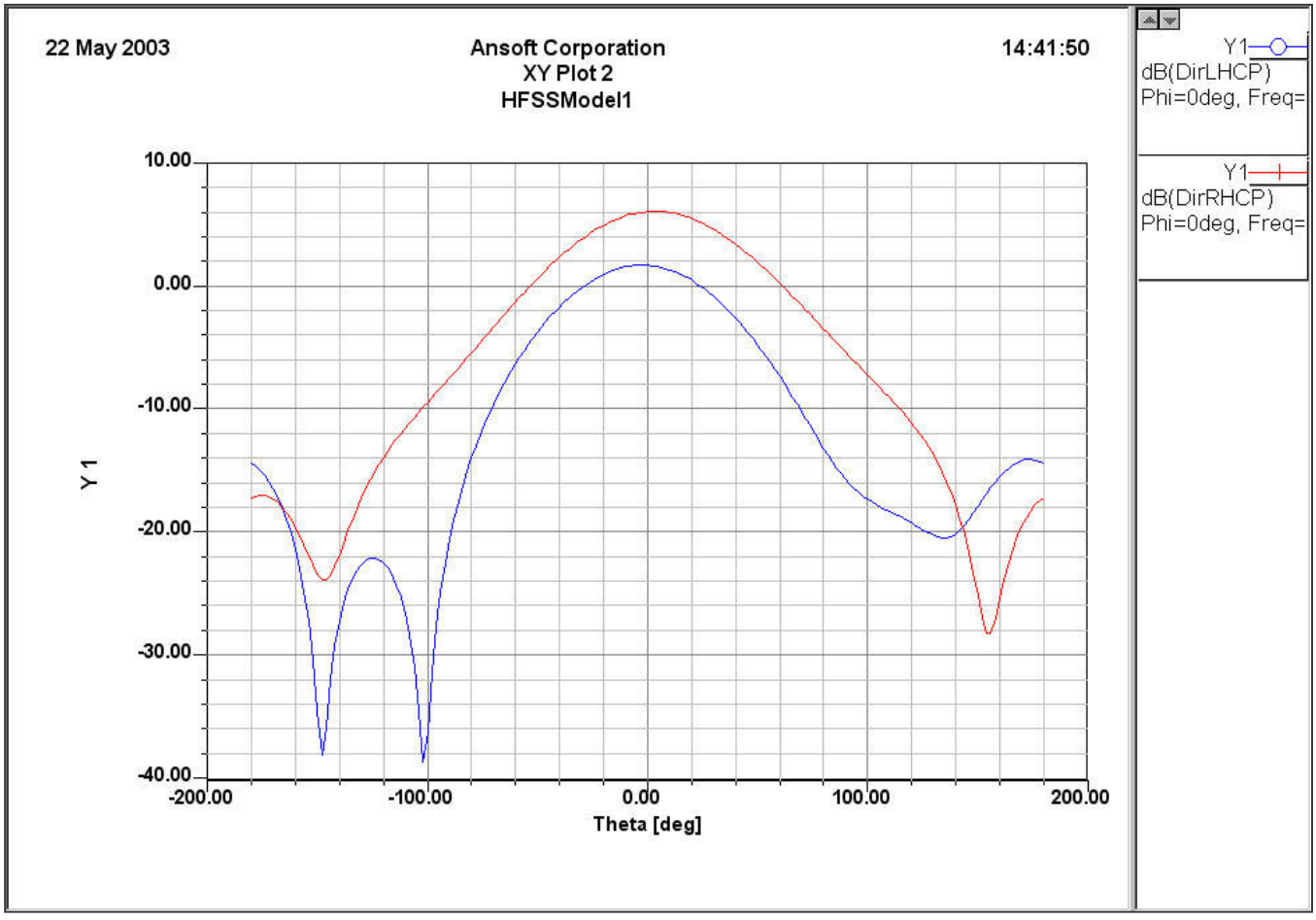
1. Select ***HFSS > Radiation > Insert Far Field Setup > Infinite Sphere***



2. Enter the values and steps for **Theta** and **Phi**
3. You can also change the coordinate system to calculate based upon a shifted or rotated coordinate system. Select the **Coordinate System** tab, and switch to the new CS.
4. You can also change the Radiation Surface over which the far fields are computed by simply switching to the **Radiation Surface** tab, and select a new surface from any that were previously defined.
5. Click **OK**

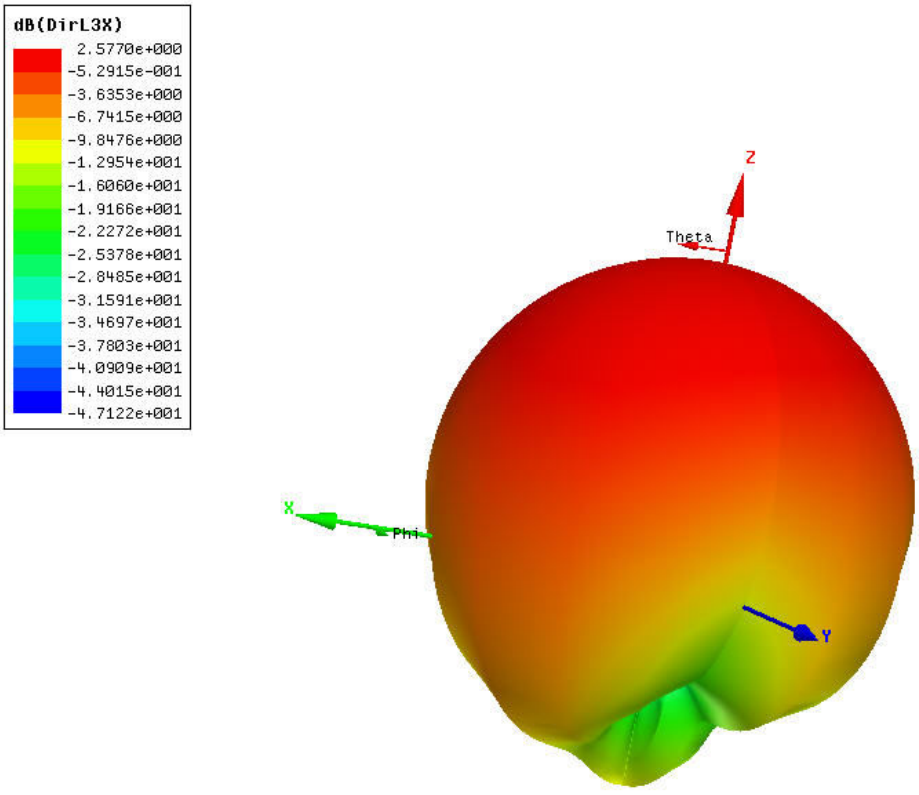
Data Plotting – Antenna Parameters

- **Creating a 2D plot:**
  1. Select ***HFSS > Results > Create Report***
  2. Select **Far Field** form the **Report Type** pulldown
  3. Select **Radiation Pattern** from the **Display Type** pulldown
  4. Select the quantity to be plotted from the **Traces** dialog
     - Note: If multiple Infinite Sphere setups exist, make sure you select the appropriate one
  5. Select **Add Trace** and **Done**
  - For a definition of the available antenna parameters, see the online help
- Below is an example of the 2D slices of a patch antenna for LHCP and RHCP directivity.



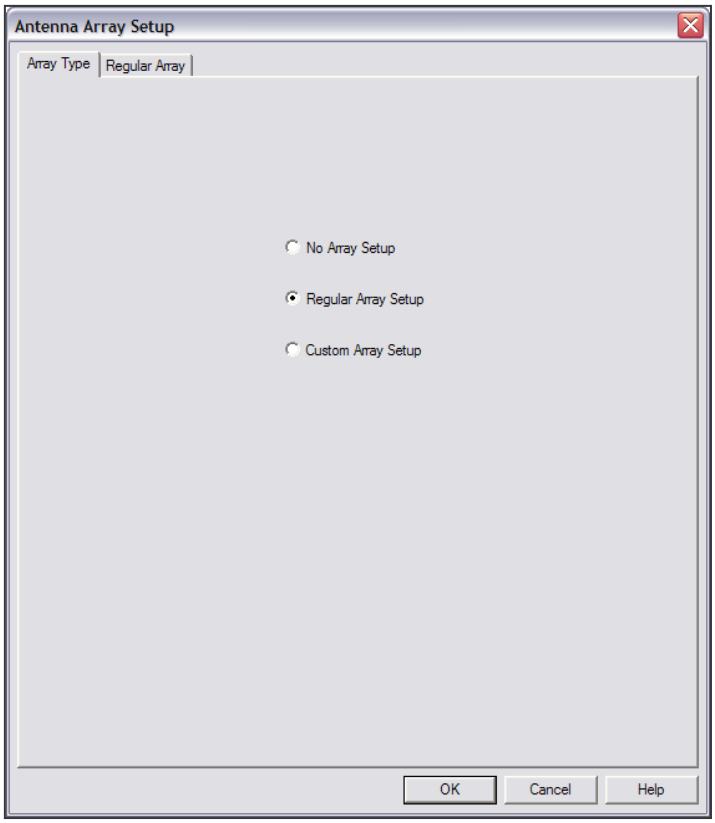
Data Plotting – Antenna Parameters

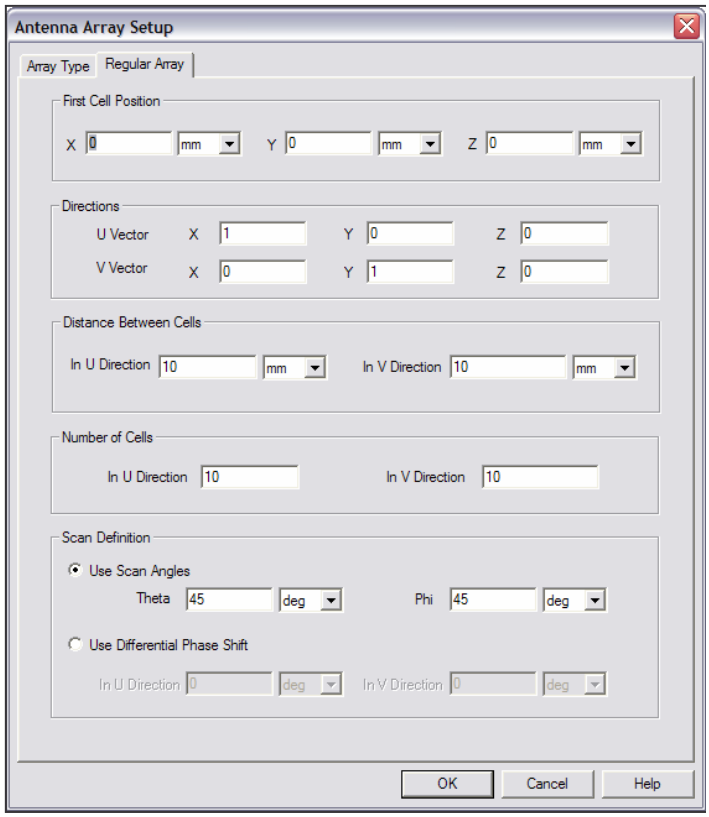
- **Creating a 3D Plot:**
  - Follow the procedures above for the 3D plot, except change the **Phi** and **Theta** quantities to match the far field calculation. Also choose an antenna quantity to plot.
- Below is an example of a 3D plot for a patch antenna:



Data Plotting – Antenna Parameters

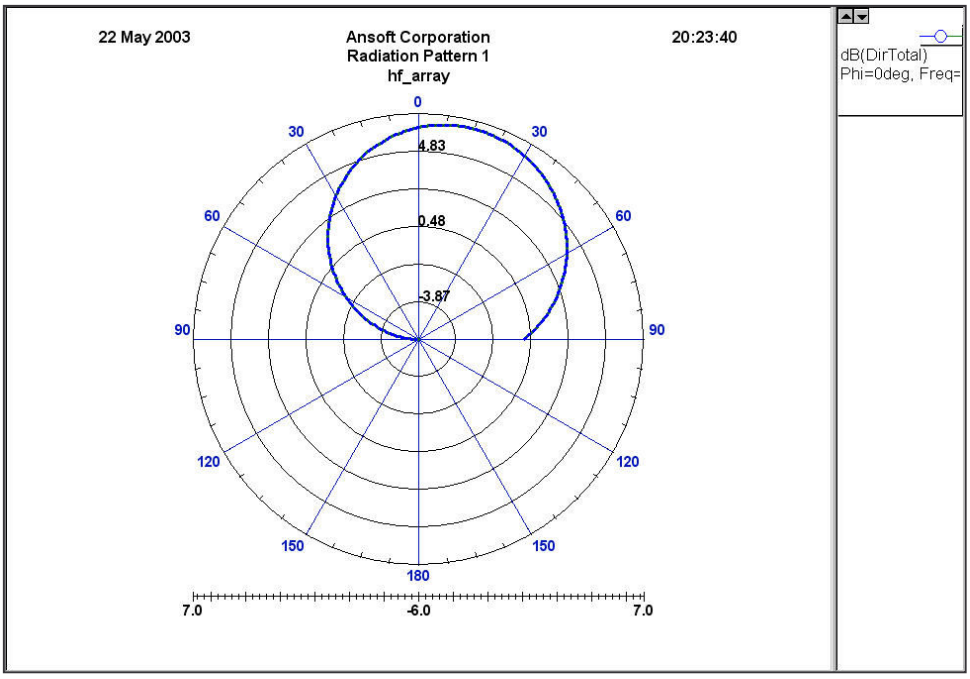
- Special Case – Antenna Arrays
  - When using master and slave boundary condition pairs to emulate an antenna array, you might want to apply an array factor to the computation of the antenna patterns. This can easily be done in Ansoft HFSS
  - **To create an Antenna Array Factor calculation:**
    1. Select ***HFSS > Radiation > Antenna Array Setup***
    2. Select **No**, **Regular**, or **Custom Array**
    3. Switch to the next tab, and enter the characteristics of the array, or enter the filename that contains the element locations
    4. Once you click OK, any plots or calculations that are displayed will be updated with the array factor calculation. To go back to the single element calculations, select **No Array Setup**

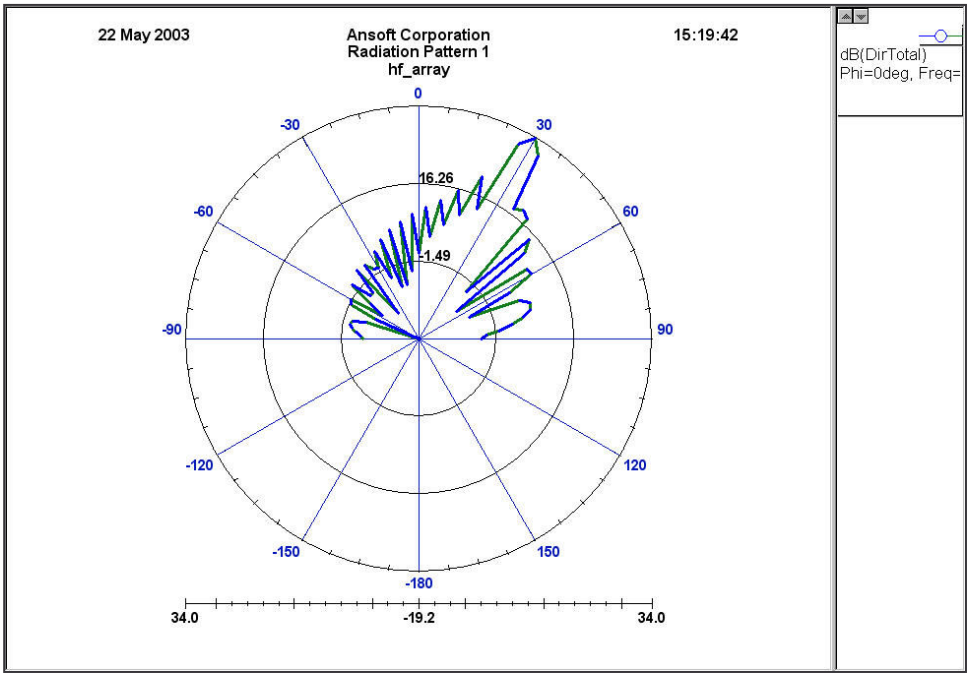




Data Plotting – Antenna Parameters

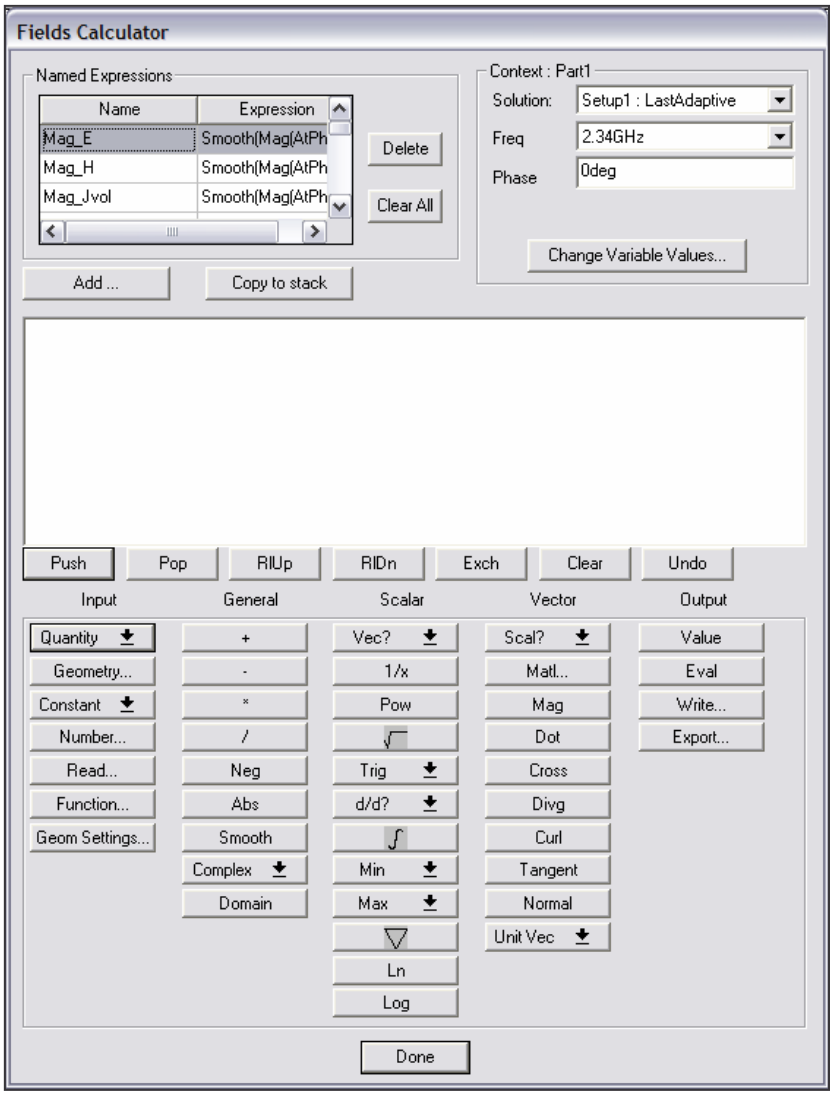
- Antenna array – Shown below are field patterns of a single element and an array setup scanned to 30 degree angle.





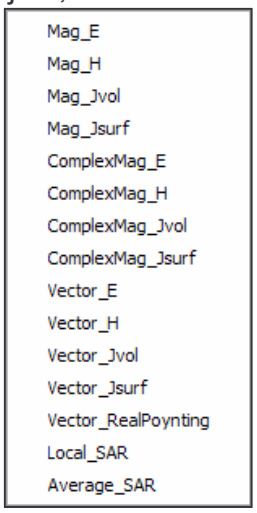
Data Manipulation

- Sometimes, the built-in calculated quantities are not adequate even given the output variable capability. Because of this, Ansoft HFSS has the flexibility built in with an arbitrary fields calculator.
- You can use the Fields Calculator to manipulate field quantities to obtain any number of values. One common use is to evaluate power flow within a structure. This can be accomplished by integrating the Poynting vector over a geometric surface. This can easily be accomplished within the field calculator.
- **To access the fields calculator:**
  - Select ***HFSS > Fields > Calculator***
- For a more detailed explanation of the functions within the field calculator, visit our online technical support at http://www.ansoft.com.ots

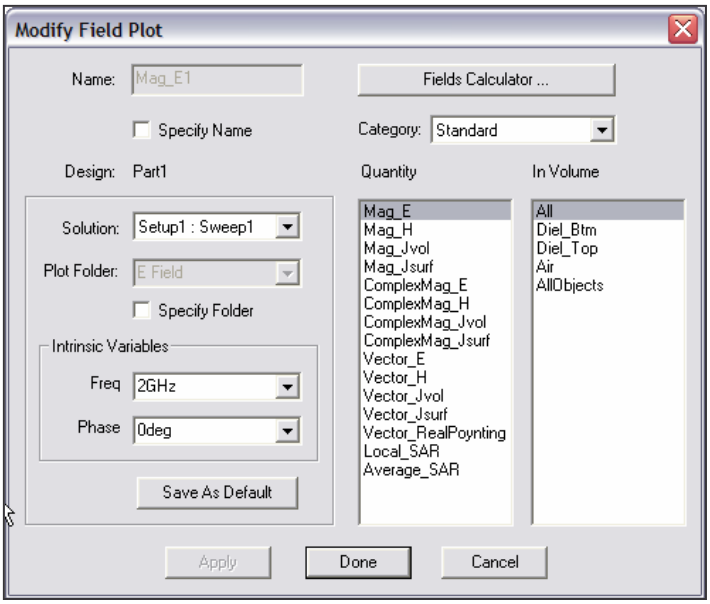


Field Overlay Plotting

- Previously mentioned was the capability to plot field quantities over a geometry within a 2D plot, but this doesn't give us a lot of insight. One of the main benefits of HFSS is that we can visualize what is going on with the electromagnetic fields within a structure, something that *cannot* be measured directly.
- Field plots can be applied to geometry selection within the 3D modeler, and can be modified given different stimulus amplitude, frequency, and phase. We can also visualize how a field propagates throughout the volume by animating this plot versus phase, which essentially adds a time base.
- **To create a 2D / 3D Field Overlay Plot:**
  1. Select a face of an object, an entire object, or even multiple object
  2. Select ***HFSS > Fields > Plot Fields >***

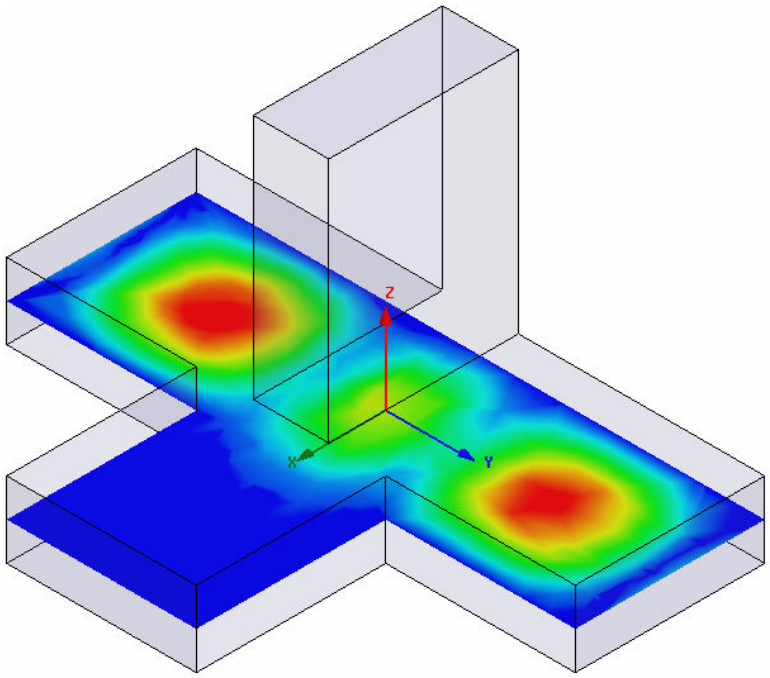


  3. You can then change the frequency of the stimulus or phase with the resultant dialog box:

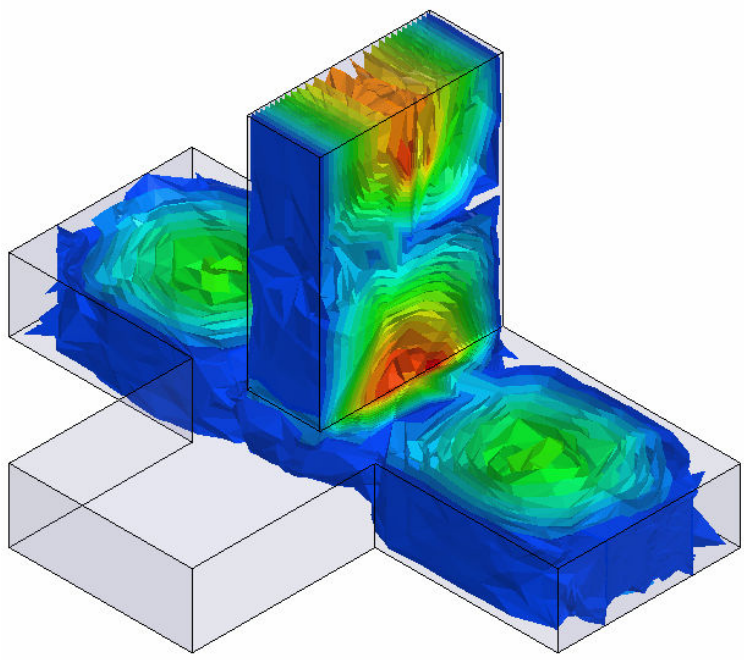


Field Overlay Plots

- Resultant modeler window for Magic “T” model with Mag_E plotted on Global:XY plane

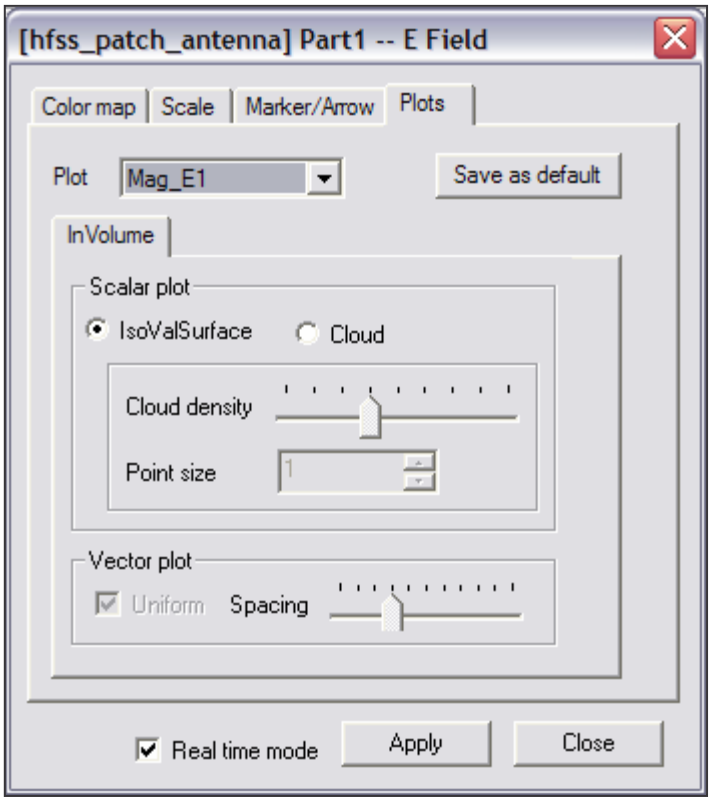


- Selecting the object “arm”, and plotting within the volume yields:



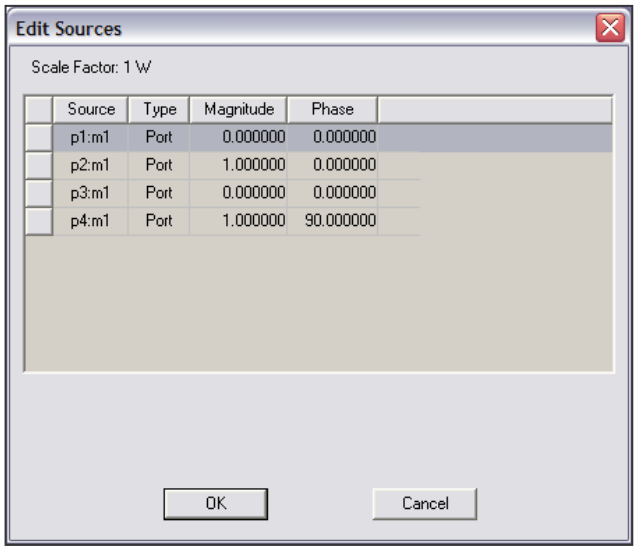
Field Overlay Plots

- The 3D plot shown on the previous page is called an Isoval surface plot. This is not the default
- **To change the field plot type:**
  1. Selecting: ***HFSS > Fields > Modify Plot Attributes***
  2. Switch to the **Plot** tab
  3. Select the applicable **Plot** from the pulldown
  4. Select the **IsoValSurface** radio button
     - You can also leave the default of **cloud plot** and adjust the **cloud density** and **point size** until the plot looks acceptable

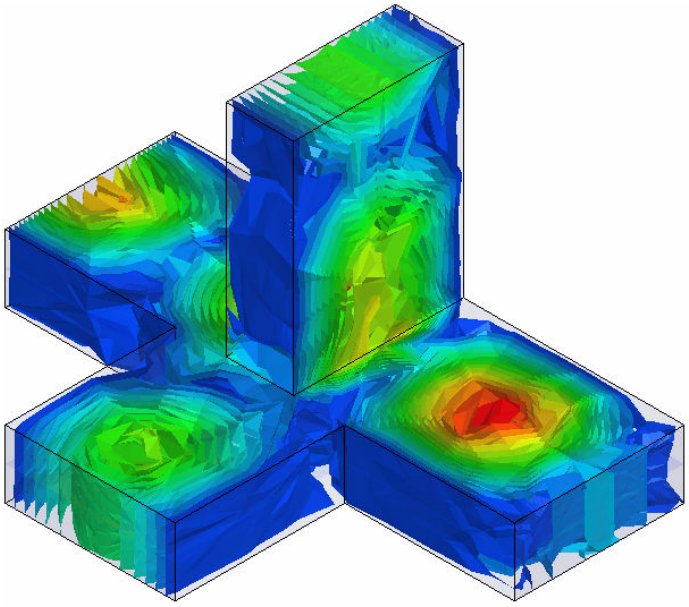


Field Overlay plots – Source Stimulus

- Sometimes it might be desirable to have multiple ports excited at the same time, perhaps out of phase. This is useful for evaluating power combiners. You might even want need to calculate the incident or reflected power for a plane wave source.
- **To change the Field Plot stimulus:**
  - Select ***HFSS > Fields > Edit Sources***

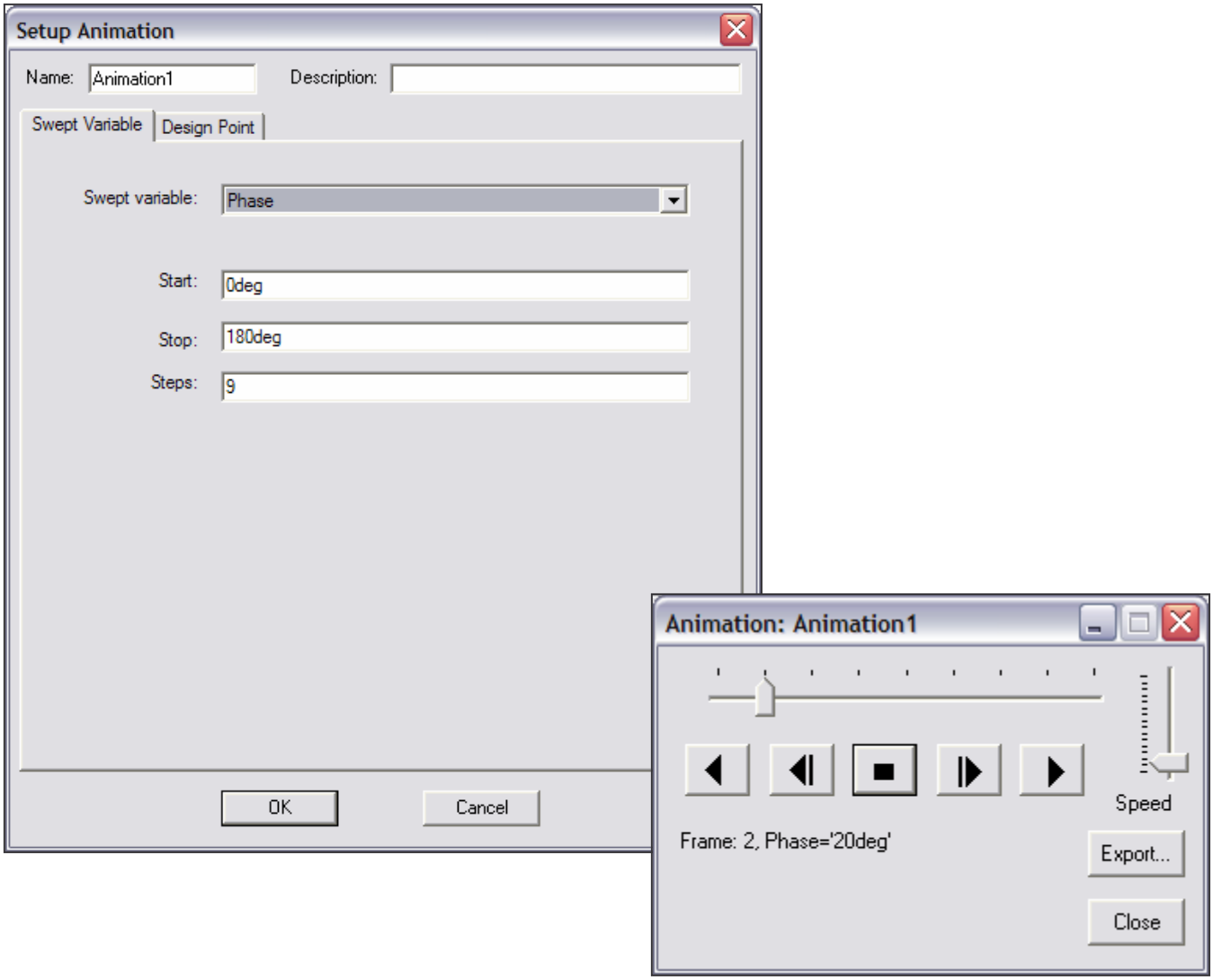


  - The following field d overlay plots shows what happens to the Magic "T" when we have a 90 degree phase shift at the 2 E-plane end inputs. The component doesn't isolate one port.



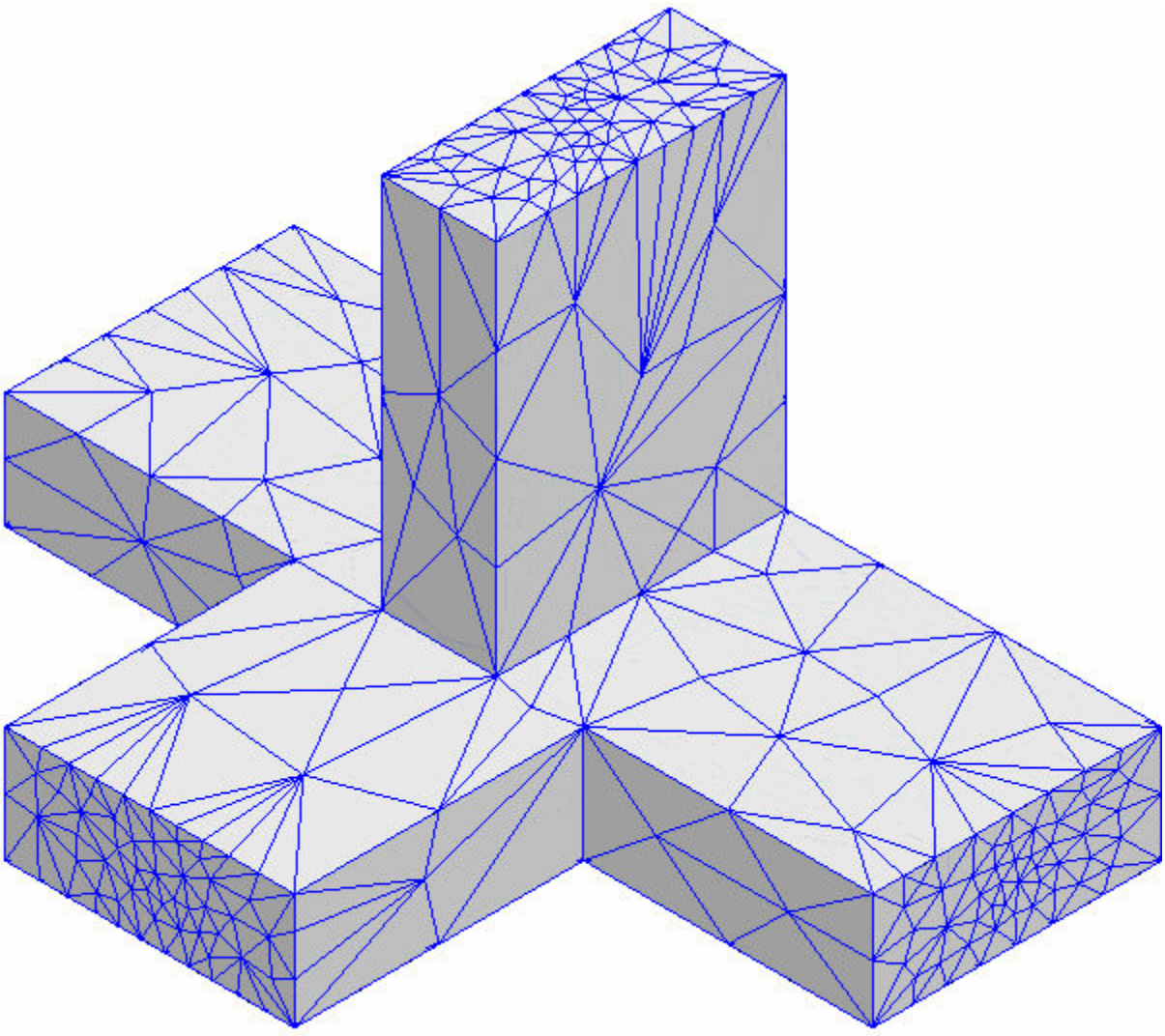
Field Overlay Plot – Phase animation

- The final step in field overlay plots is creating animation. This essentially shows you how a wave propagates through the structure as you vary phase (time).
- **To create an animated 3D field plot:**
  1. Select ***HFSS > Field > Animate***
  2. Typically, you can accept the defaults, but you may want to change the number of steps to limit computation time.



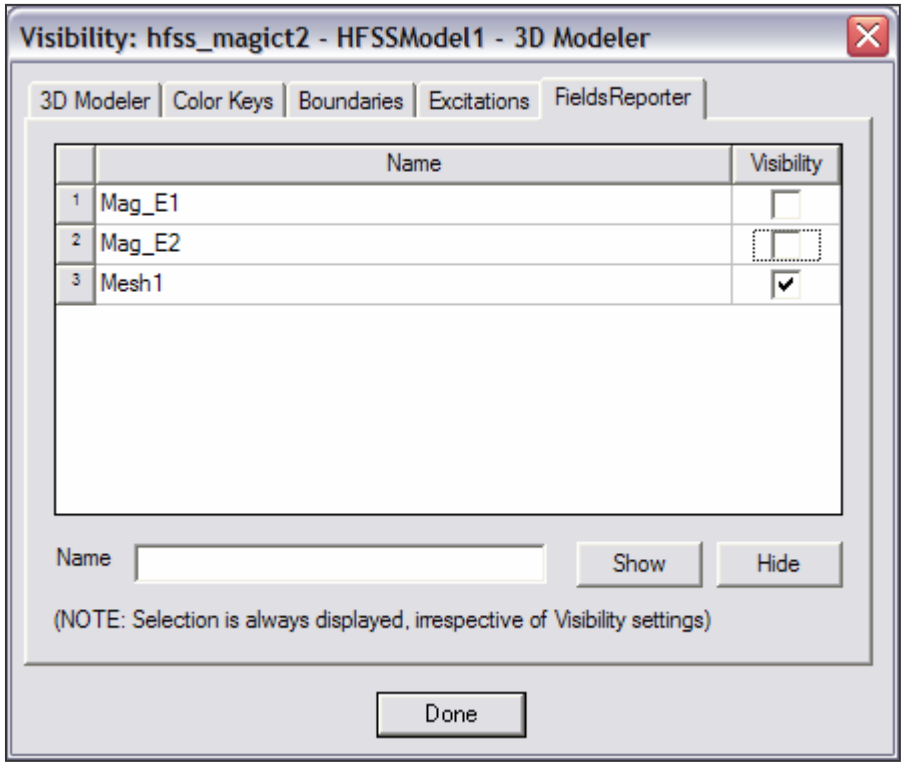
Mesh Overlay Plot

- Another plot that can be valuable in evaluating a simulation is the mesh overlay plot. Once created, you can tell which areas of the structure may need more tetrahedra for an improved quality of mesh.
- **To create a Mesh overlay plot:**
  1. Select the face, plane, or object to plot the mesh
  2. Select ***HFSS > Fields > Plot Mesh***



Viewing in the 3D modeler

- With the ability to plot field quantities and mesh within the 3D modeler, it can sometimes get crowded, and model details can be obscured by the plots. You can reduce the clutter by turning off the visibility of objects and even field plots.
- **To turn off the visibility of field overlay plots:**
  1. Select View > Visibility
  2. Select the **Fields Reporter** tab, and check, ☑, or uncheck, ☐, the fields plot you want displayed.



This page is intentionally left blank.

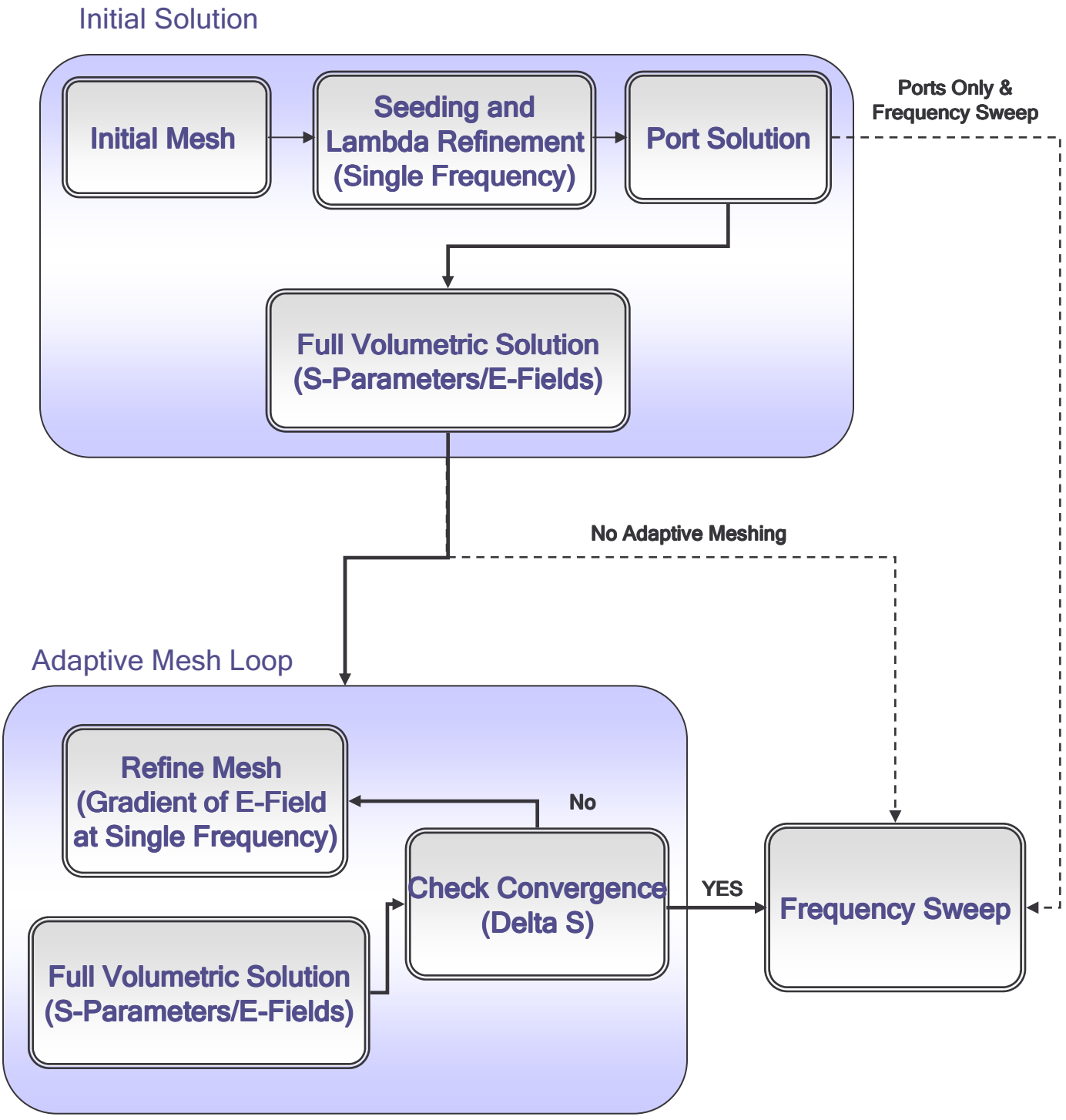
Solve Loop

- Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, HFSS will take over and step through several stages of the solution process. To start the solution process, right click on **Analysis** in the HFSS Model Tree and select **Analyze**.

- It is important to review this section since the solution setup has a direct impact on the automated solution process. A closer look at the solution process reveals it consists of three components:
  - Initial Solution – includes the mesh generation, ports solution, and a full solution at a single frequency
  - Adaptive Refinement Loop – refines the mesh and performs a full solution at the Initial solution frequency until convergence
  - Frequency Sweep – calculates the swept frequency response using a mesh generated by the Adaptive Refinement Loop

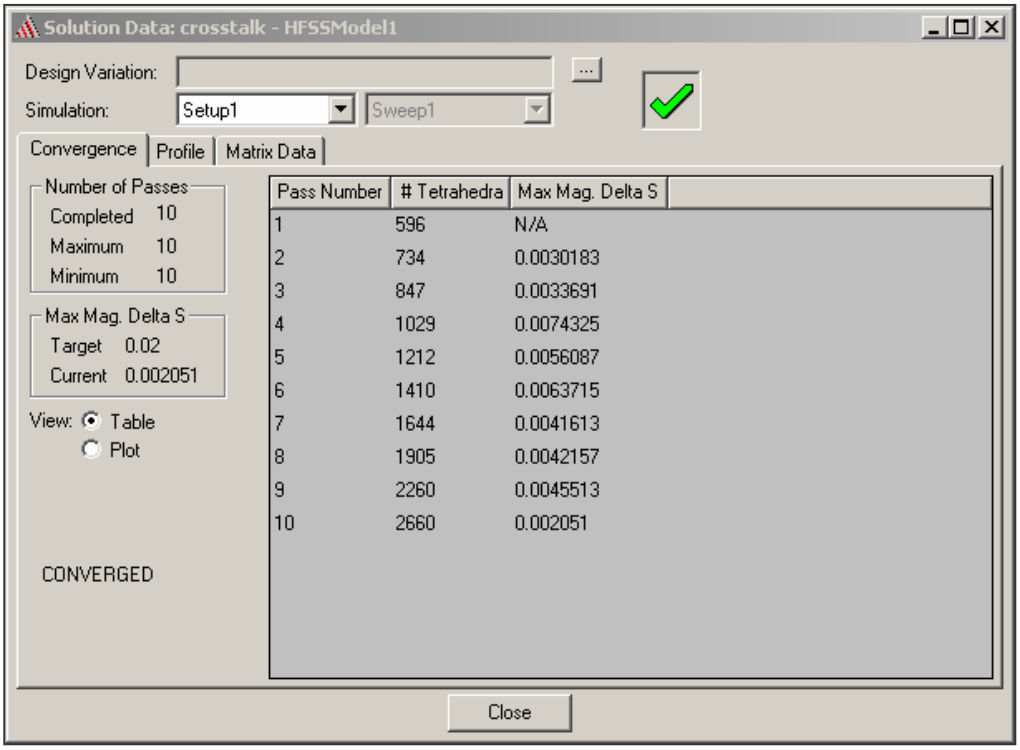
- The illustration on the following page outlines the steps performed by the solution process.

Solve Loop


Initial Solution
Initial Mesh
Seeding and Lambda Refinement (Single Frequency)
Port Solution
Ports Only & Frequency Sweep
Full Volumetric Solution (S-Parameters/E-Fields)
No Adaptive Meshing
Adaptive Mesh Loop
Refine Mesh (Gradient of E-Field at Single Frequency)
No
Check Convergence (Delta S)
YES
Frequency Sweep
Full Volumetric Solution (S-Parameters/E-Fields)

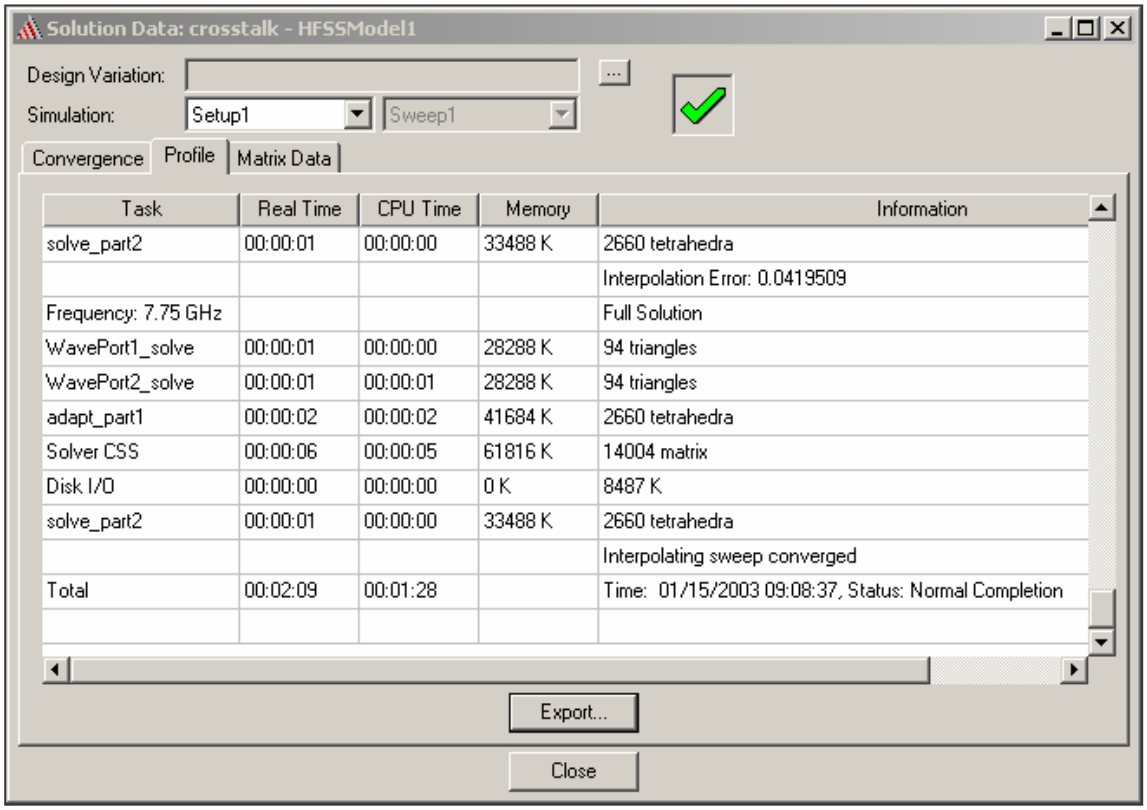
Monitoring Convergence

- You can view the Convergence of the entire solution process. Right click on **Analysis/Setup** in the HFSS model tree. The **Convergence Tab** can be used to display a table or a plot.



Profile

At any time during or after the solution process, you can examine the computing resources - or profile data - that were used by HFSS during the analysis. The profile data is essentially a log of the tasks performed by HFSS during the solution. The log indicates the length of time each task took and how much RAM/disk memory was required

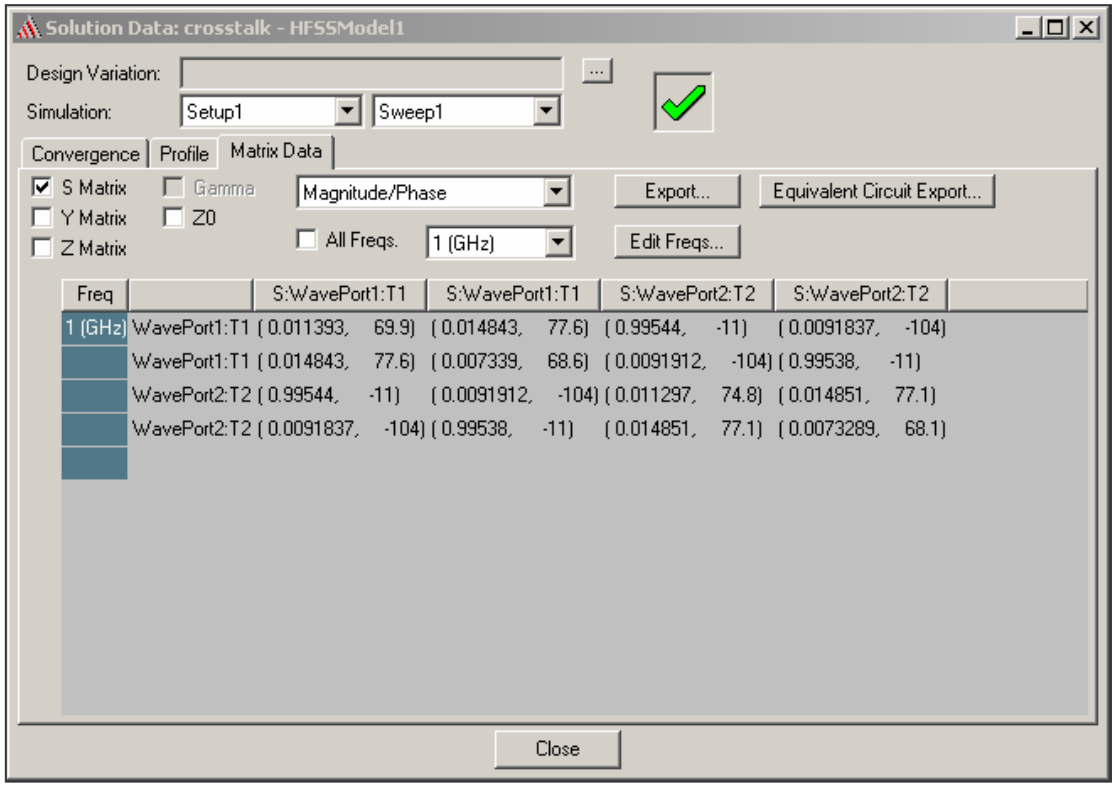


Task	Real Time	CPU Time	Memory	Information
solve_part2	00:00:01	00:00:00	33488 K	2660 tetrahedra
				Interpolation Error: 0.0419509
Frequency: 7.75 GHz				Full Solution
WavePort1_solve	00:00:01	00:00:00	28288 K	94 triangles
WavePort2_solve	00:00:01	00:00:01	28288 K	94 triangles
adapt_part1	00:00:02	00:00:02	41684 K	2660 tetrahedra
Solver CSS	00:00:06	00:00:05	61816 K	14004 matrix
Disk I/O	00:00:00	00:00:00	0 K	8487 K
solve_part2	00:00:01	00:00:00	33488 K	2660 tetrahedra
				Interpolating sweep converged
Total	00:02:09	00:01:28		Time: 01/15/2003 09:08:37, Status: Normal Completion

- Task -- lists the software module that performed a task during the solution process, and the type of task that was performed. For example, for the task *mesh3d_adapt*, Mesh3d is the software module that adaptively refined the mesh.
- Real Time -- the amount of real time required to perform the task.
- CPU Time -- the amount of CPU time required to perform the task.
- Memory -- the peak amount of memory used by your machine while performing the task. This value includes all of the applications running at the time; it is not limited to HFSS..
- Information -- general information about the solution, including the number of tetrahedra used in the mesh.

Matrix Data

- After the Solution is complete the Matrix Data can be viewed by clicking on Analysis/Setup. Right click on **Setup** and select **Matrix Data**. The **Solution Data** window will appear.



- In the Simulation pull-down list, click the solution setup and solved pass - adaptive, single frequency solution, or frequency sweep - for which you want to view matrices.
- Select the type of matrix to view.
  - S-matrix
  - Y-matrix
  - Z-matrix
  - Gamma
  - Zo (characteristic impedance.)
- The available types depend on the solution type

Matrix Data (cont.)

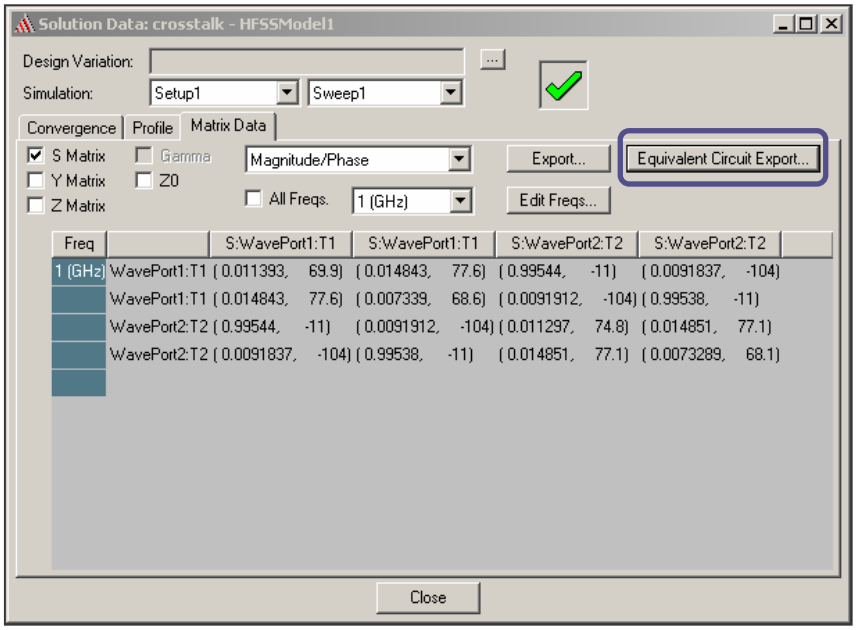
- Data can be displayed in the following formats:
  - Magnitude/ Phase
  - Real/ Imaginary
  - dB/ Phase
  - Magnitude
  - Phase
  - Real
  - Imaginary
  - dB
- The available formats depend on the matrix type being displayed.
- You can also select solved frequencies that you would like to display
  - To display the matrix entries for all solved frequencies, choose **All Freqs**.
  - To show the matrix entries for one solved frequency, clear All Frequencies and then select the solved frequency that you want to view.
  - For adaptive passes, only the solution frequency specified in the Solution Setup dialog box is available. For frequency sweeps, the entire frequency range is available.
- To insert of delete one or more displayed frequencies, click **Edit Freqs**.
- This command is only available if the sweep type is Fast or Interpolating. If you choose to export the matrix data for the Fast or Interpolating sweep after modifying the frequencies in the Edit Frequencies dialog box, only those frequencies displayed under the Matrix Data tab will be exported.

Exporting Matrix Data

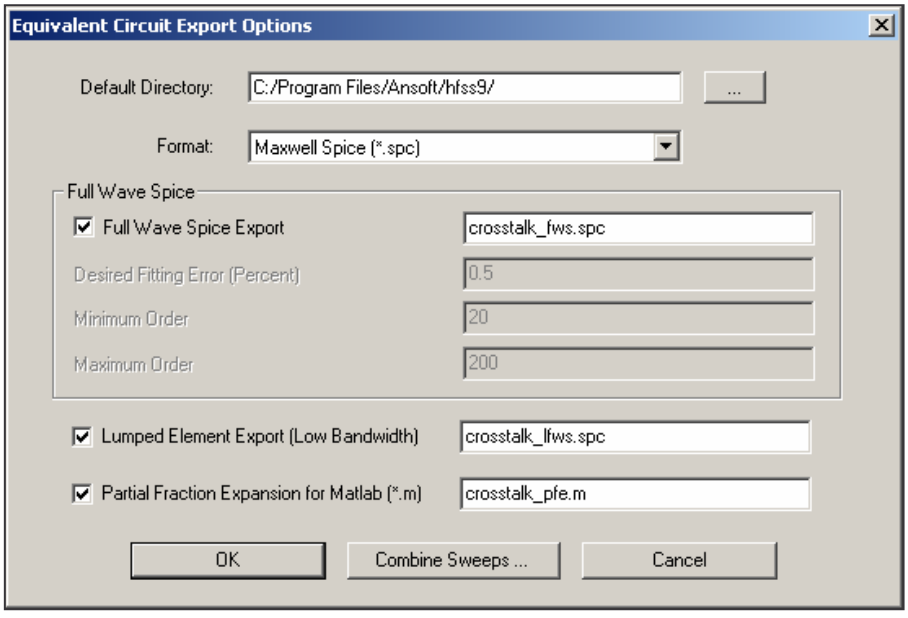
- A number of export formats are available:
  - Touchstone (*.sNp)
  - Data Table (*.tab)
  - Planar EM/HFSS v6+ (*.szg)
  - Neutral Model Format (*.nmf)
  - MATLAB (*.m)
  - Citifile (*.cit)

Equivalent Circuit Export

- It is possible to export Full-Wave Spice, Lumped Element and Partial Fraction Expansion models from Ansoft HFSS. To do this, click on **Analysis/Setup/Matrix Data/Equivalent Circuit Export**.



- The Equivalent Circuit Export Options window will appear



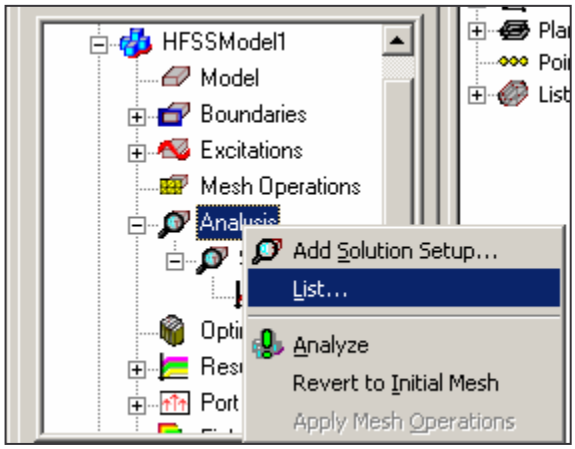
- If you have a Driven Terminal solution you can export to **PSpice, HSPICE, Spectre, or Maxwell Spice**. You must have a frequency sweep solution and five or more frequency points to successfully export an equivalent circuit data file.
- Select **Lumped Element Export (Low Bandwidth)** if you want to save the data as a low-frequency circuit model using simple lumped elements (resistors, capacitors, inductors, and dependent current sources).
- Select **Partial Fraction Expansion for Matlab** to create a *.m file

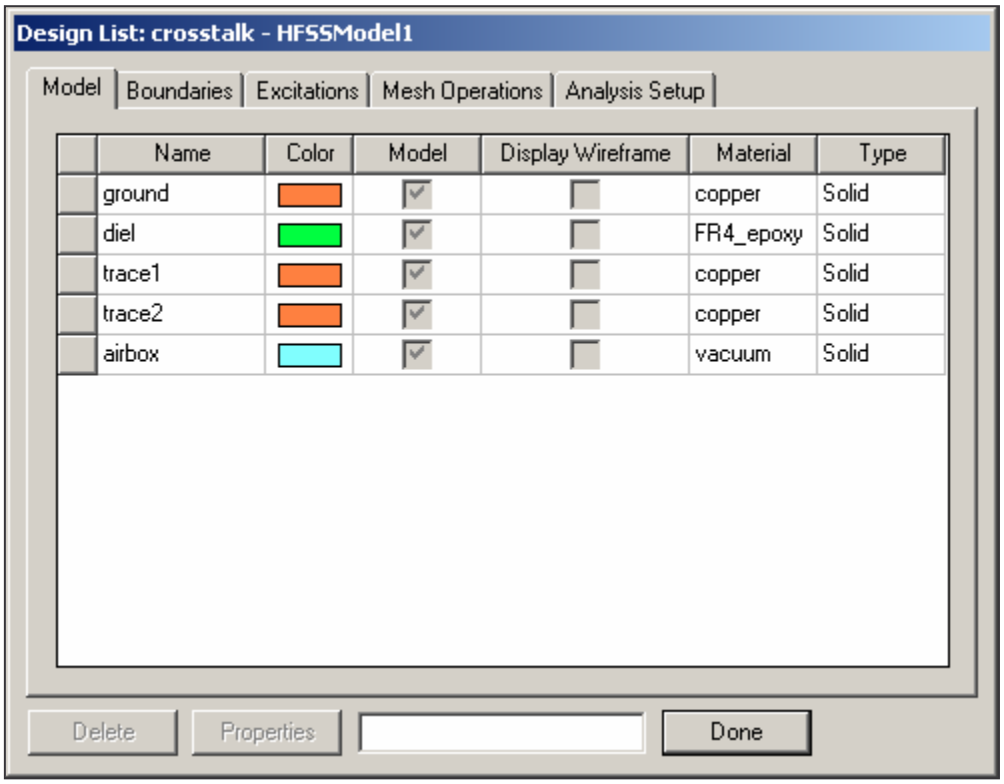
- Click **Combine Sweeps** if you want to combine the data from two or more frequency sweeps into one file. The end points of the sweep ranges can touch, but may not overlap.

- The S-matrices are written to the data file that you specified in the equivalent circuit data format.

List

- To view Model Parameters, Boundaries, Excitations, Mesh Operations and Analysis Setup right click on **Analysis** in the Ansoft HFSS model tree and select **List**





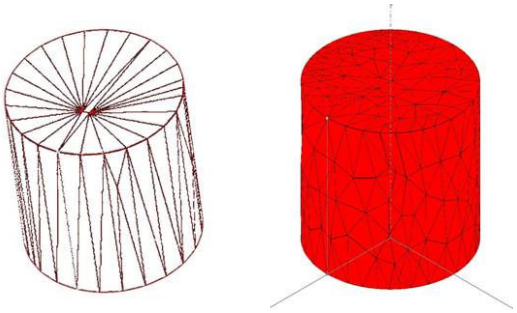
This page is intentionally left blank.

Mesh Operations

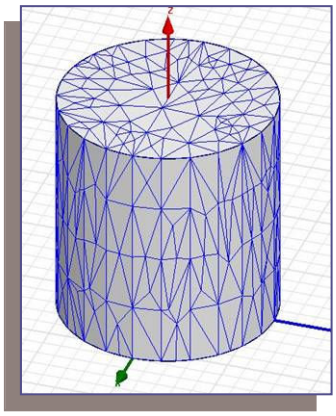
- This chapter provides details on meshing in the Ansoft HFSS v.9.0 software. It discusses the default meshing of curvature, user control thereof, and the new **volume correction** capabilities used in matrix solution. The following topics are discussed:
  - Curved Geometry Mesh Adaptation
  - Faceting Default Settings
  - Volume Perturbation in FEM Solutions
  - User-Defining Surface Approximations
  - Application Recommendations
  - Applying Mesh Operations
- **Examples/Benchmarks**
  - The following examples are provided to demonstrate the topics discussed in the chapter:
    - Standard Pillbox Resonator
    - Hemispherical Dielectric Resonator Antenna with Cavity
    - Accelerator Spoke-Cavity Application
    - Circular WG Quadrature Ortho-mode Junction (OMJ)

Curve Mesh Adaptation in HFSS v9.0

- HFSS v9.0 meshes handles curved surfaces differently than prior versions of HFSS. Proper understanding of the differences in curved mesh handling and the advantages of new capabilities in the HFSS v9.0 solver are essential to obtaining accurate results.
  - The new graphical drawing interface encourages the use of true-curved drawing, by removing the option to assign a facet count to the construction of primitives such as circles, cylinders, spheres, and ellipses.
    - Faceted primitives are available however as polyhedrons and polyhedral solids, if still desired.
  - Initial meshing is constrained by faceting decisions made by the first pass of the meshing algorithms. In prior versions of HFSS however, adaptive mesh points could be placed 'anywhere' on the true-surface of the affected object(s), as shown in the before and after images below. (Initial mesh left, adapted at right. Note that regular faceting was not maintained after adaptive mesh alteration.)

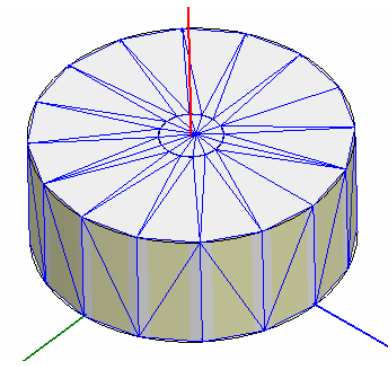


  - For HFSS v9.0, however, in order to provide more robust meshing with respect to more complex geometries, the initial faceting selections made for curved objects are respected throughout the adaptation process, so that the adapted mesh is a subdivided variation of the *same meshed volume* as the initial mesh. An adapted mesh from HFSS v9.0 is shown below.



Faceter Default Settings

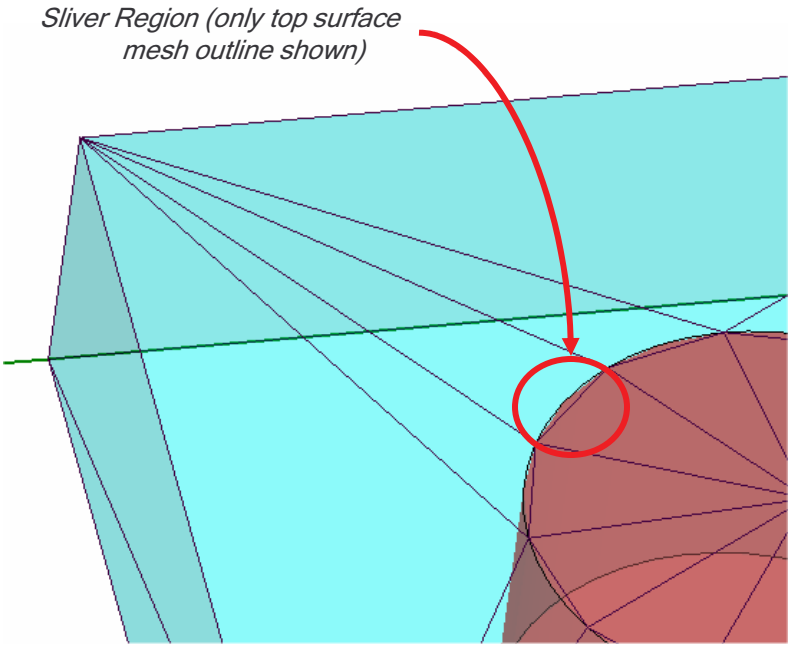
- In order to keep initial meshes at reasonable sizes, the initial faceting default setting is to constrain the mesh surface normals to fall within ***30degrees*** of the true-curved surface normals.
  - This means that a cylindrical surface would be 'faceted' into 16 segments about its circumference, as illustrated below.
  - The normal for each flat segment cannot be off by more than 30 degrees from the normal for the curved true surface which that segment's face is approximating.



  - Although aided by the **Volume Perturbation** technique now used in HFSS v9.0, it is not anticipated that this default faceting will be 'enough' for extremely high-accuracy analysis of resonant cavity volumes, spheroids, etc.
  - The faceting default does not include a stringent aspect ratio default (note long triangles on top and bottom cylindrical faces).
- The default however is sufficient for most applications in which the curved geometry is not itself highly critical to the simulation result, yet where more faceting would result in an extremely large initial mesh
  - E.g. Applicable for:
    - …meshing of transmission line models with both signal and non-signal vias present to prevent parallel-plate modes in stripline or CPW configurations
    - …Coaxial applications, where both the inner and outer radius faceting maintains an effectively congruent distance for characteristic impedance results.
    - …cylinders or spheres used as radiation boundaries.
  - Less Applicable for:
    - ….cylindrical cavity resonances to 0.001% frequency accuracy
    - …elliptical waveguide coupling irises
    - …circular waveguides

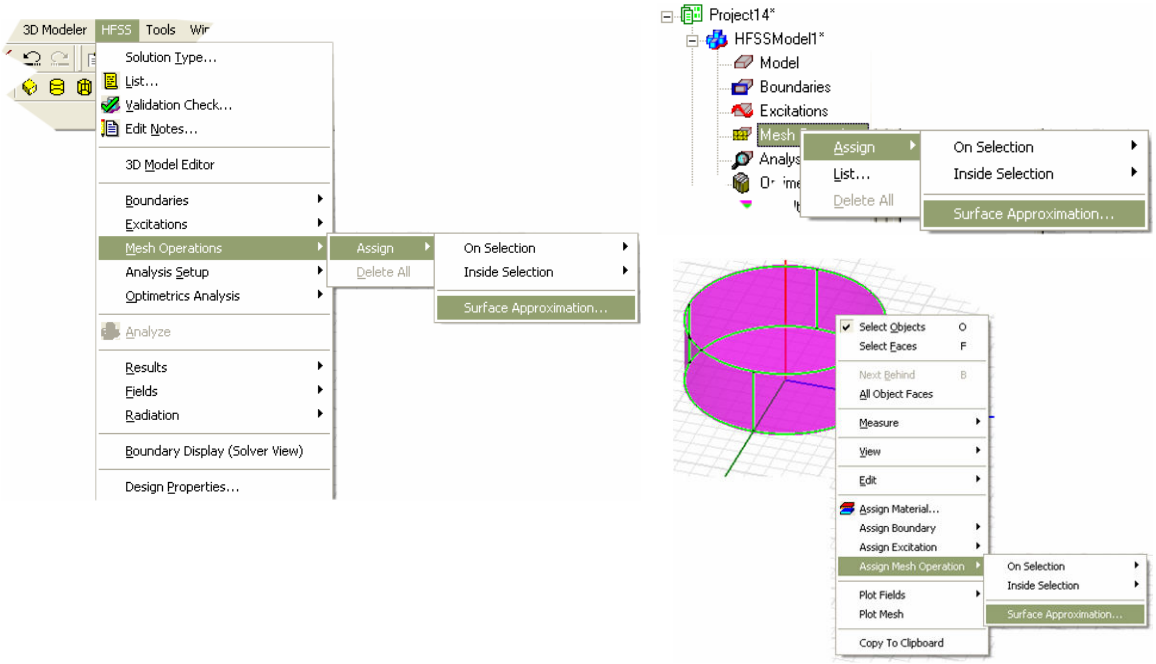
Introduction of Volume Perturbation to HFSS Solutions

- Despite lower faceting 'compliance' to the true-curved geometry as the adaptation process adds mesh points, HFSS v9.0 introduces a new solution technology that should result in a more accurate solution from far smaller mesh counts than in prior versions. This technology is referred to as **Volume Perturbation** or **Volume Correction.**
  - In a faceted mesh of a cylindrical solid, small arc-section volumes are 'lost' to the cylinder volume and become part of the volume of the cylinder's surroundings.
  - Since the true geometry definition is known before faceting, the delta-volumes represented by each of these sliver regions is computable
  - The meshing algorithm in HFSS v9.0 can provide 'adjustment factors' to both the tetrahedral center-edge node locations and to the appropriate volumes of the tetrahedra on either side of the curved boundary, so that the FEM solution proceeds with the right terms even for a loosely-discretized mesh
  - The end result is higher accuracy from the same mesh, even before the benefits of adaptive mesh refinement are taken into account.
- Volume Perturbation solutions are always on. No user setting is required to active them.

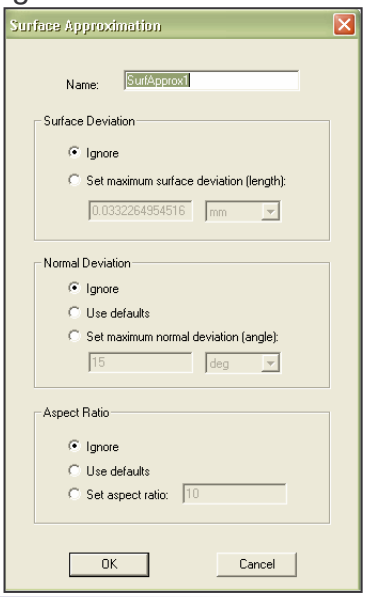


User Control of Curved Mesh Faceting

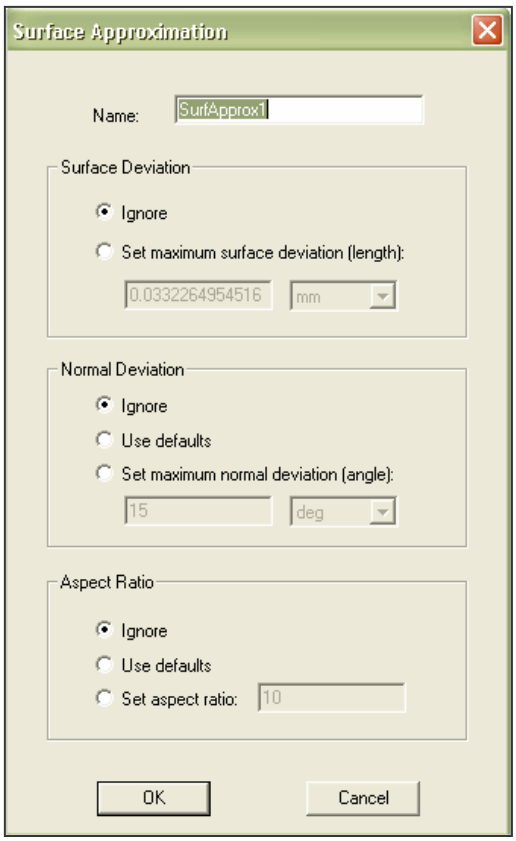
- Since Volume Perturbation alone is not always enough, users can control the fidelity to which the initial mesh faceting conforms to geometric curvature by assigning Mesh Operation Surface Approximations to appropriate objects and/or object faces
  - Mesh Operations can be assigned from the ***HFSS*** menu, from the **Design Tree**, or from the geometry interface's context-sensitive menu.



- The Surface Approximation options are shown below right. Definitions follow:
  - *Surface Deviation* is the maximum spacing, in drawing units, that the tetrahedral surfaces may be from the true-curved geometry's surface.
  - *Normal Deviation* is the maximum angular difference, in degrees, that a tetrahedral face's normal can have from the surface normal for the true geometry which it is meant to represent.
  - *Aspect Ratio* refers to the maximum allowed aspect ratio of all faces of all tetrahedra of the selected object or face. This setting influences mesh quality rather than actual meshed volume or surface locations.



- Each Setting can be forced, left at the default, or turned off entirely
  - *Ignore* means no evaluation of this constraint will be done in generation of the initial meshing, not even to the "built-in" settings HFSS follows without surface approximation instructions
  - *Use Defaults* leaves that constraint at the built-in settings, and is selected if you do not wish to tighten this constraint but also do not wish to deactivate it.
  - *Set...* is of course used to apply a user constraint value which may be tighter **or** looser than the built-in constraint used in the absence of specific instructions.
    - Setting looser constraints than the default may require also setting adjoining objects to the same looser constraint so that the default does not take precedence at interfaces.



Usage Suggestions

- **Do not overspecify.**
  - It is always easier to 'add' than subtract mesh, by running more adaptive passes or by adding supplemental mesh instructions
  - Too stringent a setting (e.g. Normal Deviation of 1 degree) can result in poor mesh qualities due to aspect ratios, poor mesh gradients to surrounding objects, etc.
- Use Aspect Ratio settings along with Normal or Surface Deviation settings
  - For cylindrical type objects where curved and planar faces meet, the normal and surface deviation settings apply to the curved faces only. Setting an aspect ratio limit as well (e.g. 4:1) will force a few additional triangles on the planar faces and help preserve a cleaner overall mesh
- Consider using Polyhedrons or Polygons instead if using to 'reduce' mesh
  - If your design has many curved objects which you want only very coarsely meshed (e.g. a whole fence of ground vias, for which 30 degree default normal deviation is unnecessary), and the geometry is not imported, consider drawing the vias as hexagonal or even square solids instead, rather than having to 'remember' to reduce the meshing fidelity on them all.

Applying Mesh Operations

- If you want to refine the mesh on a face or volume but don't want to generate a solution, do the following after defining mesh operations:
  - If a current mesh has been generated, HFSS will refine it using the defined mesh operations.
  - If a current mesh has not been generated, HFSS will apply the mesh operations to the initial mesh.
  - If an initial mesh has not been generated, HFSS will generate it and apply the mesh operations to the initial mesh.
  - If the defined mesh operations have been applied to the selected face or object, the current mesh will not be altered.
  - Define a new mesh operation rather than modify an existing mesh operation. HFSS will not re-apply a modified mesh operation. Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them.

- Source: (Canonical)
- Description
  - A pillbox resonator is a simple cylindrical cavity, the exact resonance of which is solved analytically as:

$$f_r^{TE_{mnp}} = \frac{c_\varepsilon}{2}\sqrt{\left[\frac{q_{nm}^2}{(\pi b)^2} + \frac{p^2}{l^2}\right]}$$

$$f_r^{TM_{mnp}} = \frac{c_\varepsilon}{2}\sqrt{\left[\frac{p_{nm}^2}{(\pi b)^2} + \frac{p^2}{l^2}\right]}$$

    - Equations for TE and TM modes of order (n, m, p) where p is the number of half-wave variations in the cavity axis direction
    - p_{nm} and q_{nm} represent the zeros of the Bessel functions or their derivatives
    - b is cavity radius, and l is cavity height
    - c_ε is the speed of light in the enclosed volume media
- Benchmark
  - A pillbox project with b = 10 cm and l = 10 cm was constructed.
  - Volume of cavity was vacuum; wall boundaries were perfect conductors
  - Solution was requested for 3 eigenmodes, with Surface Approximations set to vary the Normal Deviation
    - *norm* was varied from 5 to 45 degrees by 5 degree steps
    - *Aspect ratio* was set at 5:1; Surface Deviation setting was “Ignore”
    - Solution setup was 0.25 GHz starting frequency (to assure a starting frequency did not add mesh beyond those set by the surface approximation settings
    - Solutions were continued to 0.01% delta-f (real part) or 10 passes, whichever came first
  - Tabulated results vs. theoretical computations are shown on the page following
    - Bessel zeros were computed to about 5 digits (e.g. p_{11} = 1.84118)
    - Speed of light used was 299792458 m/s

Results:

Normal Deviation (deg)	HFSS TM010 Result (GHz)	Delta vs. Theory (%)	HFSS TE111 Result 1 (GHz)	HFSS TE111 Result 2 (GHz)	Avg. Delta vs. Theory (%)	Final Tetrahedra Count	Final Delta-f Convergence (%)
5	1.147406	0.001482	1.7361	1.736118	0.0756005	3347	0.0067755
10	1.147345	0.006798	1.73307	1.733154	0.2480399	2670	0.0077428
15	1.147124	0.026058	1.72758	1.727638	0.5647448	1882	0.022301
20	1.146748	0.058827	1.7192	1.719607	1.0370537	1060	0.042848
25	1.146623	0.069721	1.71165	1.712348	1.4632887	1087	0.040982
30	1.146196	0.106935	1.69707	1.697355	2.3142622	1008	0.052023
35	1.145602	0.158703	1.68868	1.688798	2.8019963	890	0.12154
40	1.144962	0.214481	1.67654	1.677916	3.4645009	887	0.033112
45	1.144962	0.214481	1.67654	1.677916	3.4645009	887	0.033112

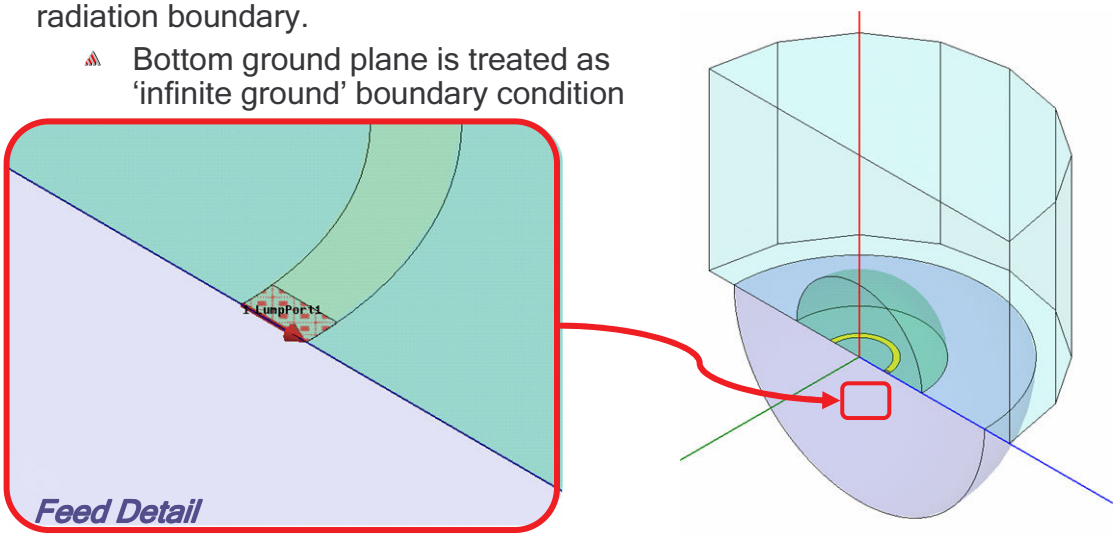
Observations:

- With higher-order modes, absolute error is higher in order of ascending frequency regardless of normal deviation faceting applied
  - This is expected due to some solution dependence on ‘starting frequency.
  - Added adaptive elements are ‘shared’ among modes. Yet higher order modes have more ‘focused’ field peaks as compared to more fundamental modes. Therefore more rapid improvement for lower order modes are an artifact of the adaptation technique and not one volume correction can strongly influence.
  - Note that for higher-accuracy analysis of the higher frequency modes, the solution can be executed with solution settings that will find that as the lowest frequency mode.
  - Even 22.5 degree setting will result in better than 0.05% error for the fundamental mode solution.
- 40 and 45 degree cases had identical meshes and results
  - When used to ‘reduce’ mesh, other quality checks may still constrain the result to more than just the surface approximation settings requested.

Conclusions:

- For simple canonical shapes and fundamental modes, increased faceting is unnecessary unless extremely tight accuracy deviations are required

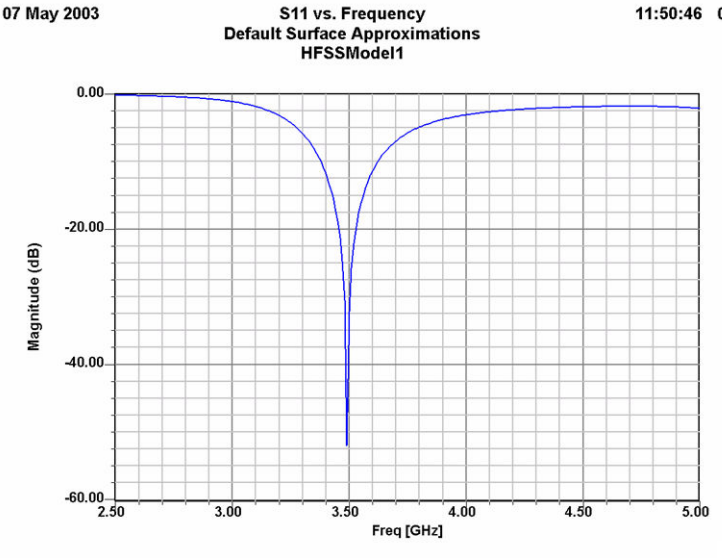
- ## Source:
  - K. K. So and K. W. Leung, “Annular Slot Excited Dielectric Resonator Antenna with a Backing Cavity,” Proceedings of the 2002 IEEE Antennas and Propagation Society International Symposium, San Antonio, TX, June 2002, Volume 4
- ## Description:
  - A hemispherical dielectric resonator antenna (DRA) is fed by a coaxial cable across an annular ring slot, which couples the DRA to a hemispherical cavity beneath it. This project has a combination of spherical and planar-circle curves which should give good evidence of the utility of the volume correction approach. The source provides both author-derived analytical results and measurement results.
- ## Benchmark:
  - An image of the modeled geometry is shown (below, right).
    - Symmetry was used (1 plane).
    - Cavity (vacuum filled) is 25mm radius; DRA is 12.5 mm radius. DRA material has $\varepsilon_r = 9.5$
    - Annular ring outer radius is 5.8 mm, 1 mm width
    - Feed is generated using a lumped gap source port spanning the annular ring (see detail image)
    - A 30mm radius by 35mm height polyhedron (16 sides) is used for the radiation boundary.
      - Bottom ground plane is treated as ‘infinite ground’ boundary condition

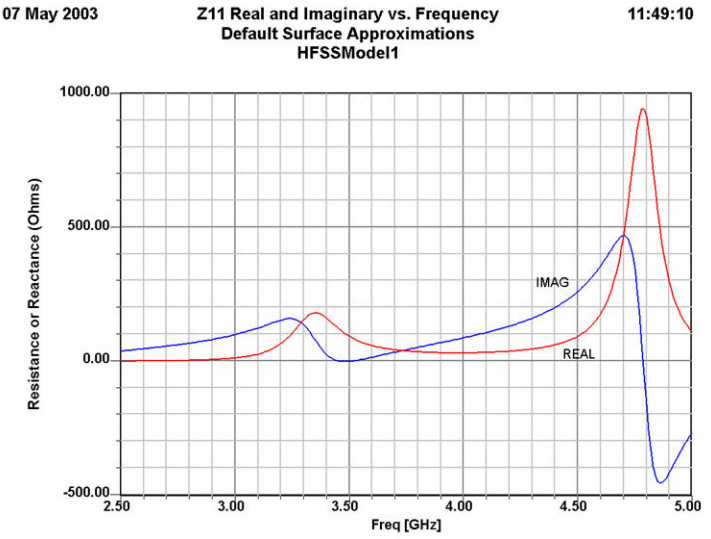


    - Solution requested using only *default* surface approximations
      - 10 adaptive passes or to delta-S of 0.01 at adaptive frequency of 3.75 GHz
      - 2.5 - 5 GHz fast sweep requested

Results:

- Plots of S11 vs. frequency and real and imaginary Z11 vs. frequency are shown
- Results conformed almost exactly to source's reported measured results, and were closer to measurements than source analysis
- HFSS v8.5 solved to effectively the same result, but required about 4 times the mesh
  - 3324 at pass 8 (v9) vs. 13,300 tetrahedra at pass 11 (v8.5)

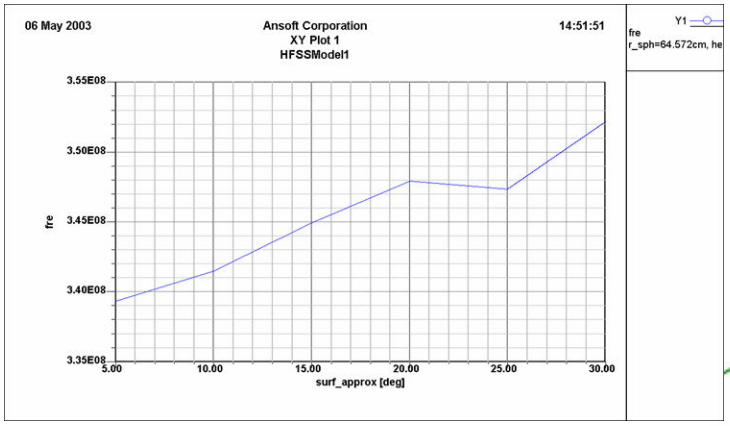


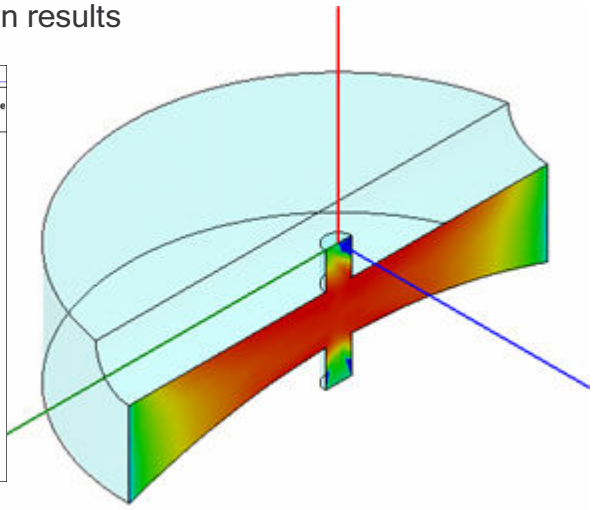


Conclusions:

- This provides an example of a project for which the volume perturbation is extremely well-suited, by providing the proper dielectric resonance volume
  - Past experiments with as many as 24 facets per circumference without volume perturbation demonstrated a noticable frequency error, therefore the role of the new solution capability in this success is clear.
- Solution accuracy was excellent in absolute terms even with default settings, and greatly exceeded HFSS v8.5's capabilities
  - A second simulation with Normal Deviation set to 15 degrees solved to the same results and excellent convergence, still beating the HFSS v8.5 mesh required

- ## Source:
  - http://www.jlab.org/~piller/spoke/spoke.htm
- ## Description:
  - A Spoke cavity is used in accelerators for ion beams. This variant is a 'low Beta' type. The cavity consists of a cylindrical volume with spherical sections subtracted from the flat ends. The 'spoke' is a smaller cylinder with its axis orthogonal to the main cavity's axis that cuts through the cavity centerline on a diameter. There is an input/output beam pipe that runs coaxial with the main cavity and exits the two concave spheroid ends. Due to symmetry these cavities are usually analyzed in 1/8 sections with PHC boundary conditions
- ## Benchmark:
  - An image of the modeled cavity geometry from the HFSS graphical interface is shown (below, right).
    - Main cavity 21.908 cm radius by 25.4cm height
    - Spherical subtractions have radius 64.572 cm
    - Spoke is 3.81 cm radius; beam pipe 1.359 cm radius
    - Solved for Normal Deviations from 5 to 30 degrees (as parametric sweep)
      - Solution settings 10 passes or to delta-f of 0.01%(real)
      - The beam ports were left as PHC bondaries, so port loading is *not* included in these simulation results





- ## Results:
  - A plot of solved resonant frequency vs. the Normal Deviation surface approximation setting used is shown above left. Tabulated results are on the page following, with mesh and error statistics
  - The reference website indicates that a different solution method for the cavity obtained a resonance frequency of 338.048 MHz, yet the measurement obtained 348.578 MHz. Deltas to both are shown in the table.

Results, cont.

Normal Deviation (deg)	f_r (MHz)	Delta vs. Ref. Analysis (%)	Delta vs. Ref. Measurement (%)	Final Tetrahedra Count	Final Delta-f Convergence (%)
5	339.332	0.379827717	2.652490978	21160	0.0021053
10	341.486	1.017015335	2.034551808	9425	0.011095
15	344.907	2.029001799	1.053135884	4246	0.10175
20	347.903	2.915266471	0.19364389	3356	0.1472
25	347.353	2.752567683	0.351427801	3757	0.016916
30	352.195	4.184908652	1.037644372	2601	0.3341

Observations:

- Since the reference analysis and measurement do not match, neither can be taken as an absolute accuracy validator for comparison to the HFSS results.
  - The measurement was of a cavity with port loading at the beam pipe(s) and a probe loop which will themselves perturb results slightly
- The trend of resonant frequency shift with tighter normal deviation restriction is quite nearly linear
- Tighter initial faceting results in better reported convergence for the same pass count
- V8.5 convergence was still not at low levels after the last solved pass. Therefore it was only coincidentally near the most-refined HFSS v9 solution. Carrying the HFSS v8.5 solution to good convergence rather than a limited number of tetrahedra took more mesh than reported for HFSS v9.

Conclusions:

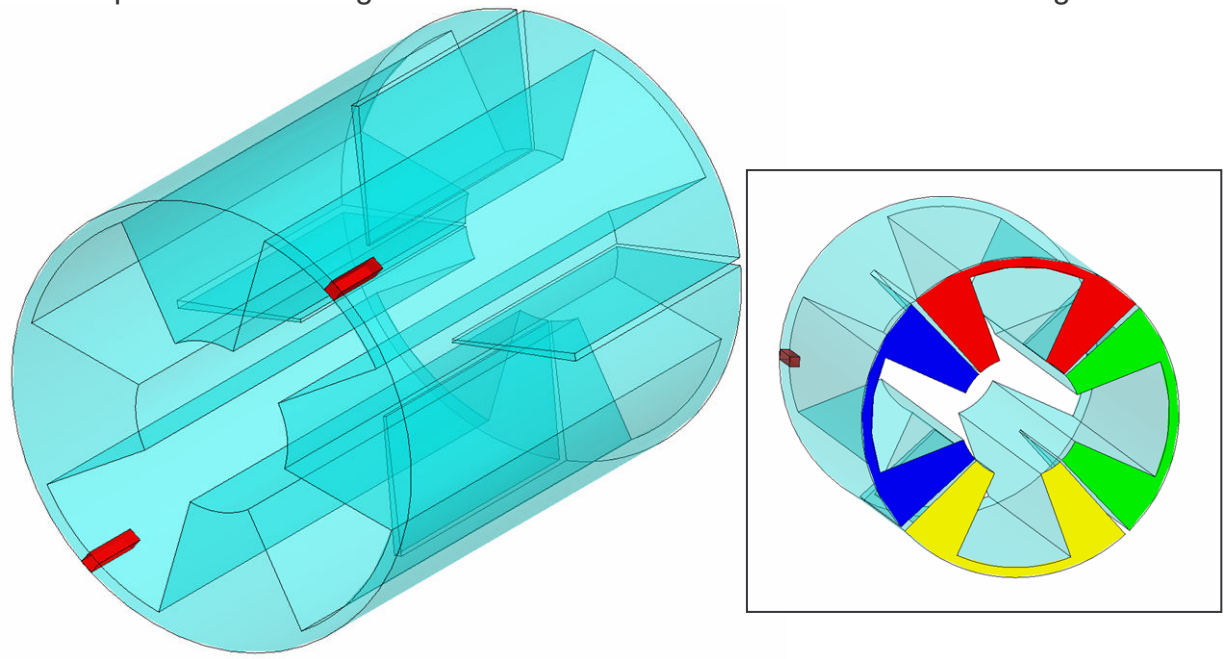
- None of the meshes resulted in inadequate volume filling or clearly incorrect solutions. This is a geometry type for which increased faceting merely fine-tunes the resulting convergence and final solution outputs.

Source:

- Henry Z. Zhang, “A Wideband Orthogonal-Mode Junction using Ridged Sectoral Waveguides,” Proceedings of the 2002 IEEE Antennas and Propagation Society International Symposium, San Antonio, TX, June 2002, Volume 4

Description:

- This quadrature orthomode junction (OMJ) uses a tapered septum to convert a circular, coaxial, quad-ridge waveguide into four contoured ridge waveguides which have modes analagous to rectangular single-ridge waveguide. The use of tapered septums intersecting with circular cross-sections necessitates drawing in true

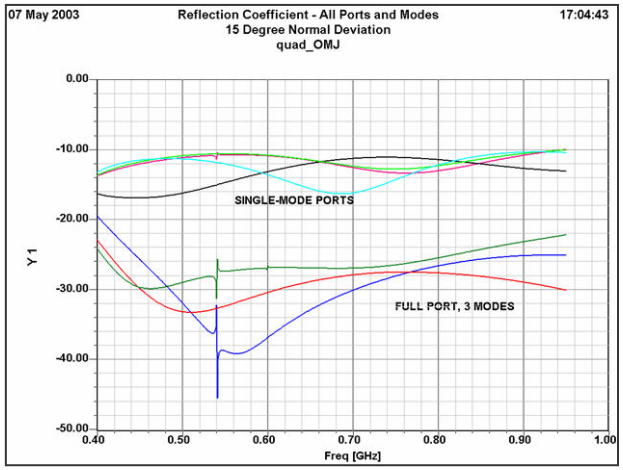


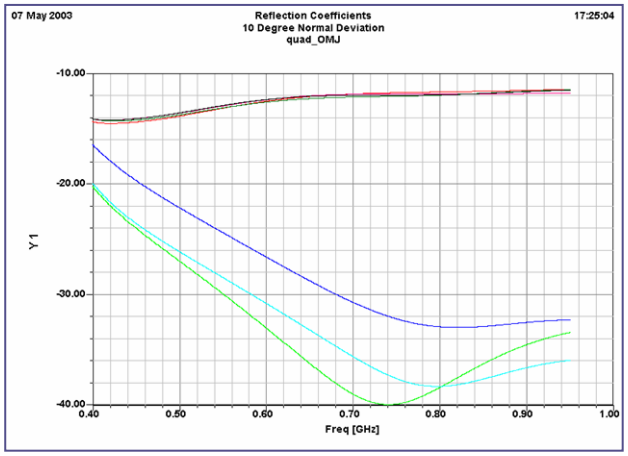
Benchmark:

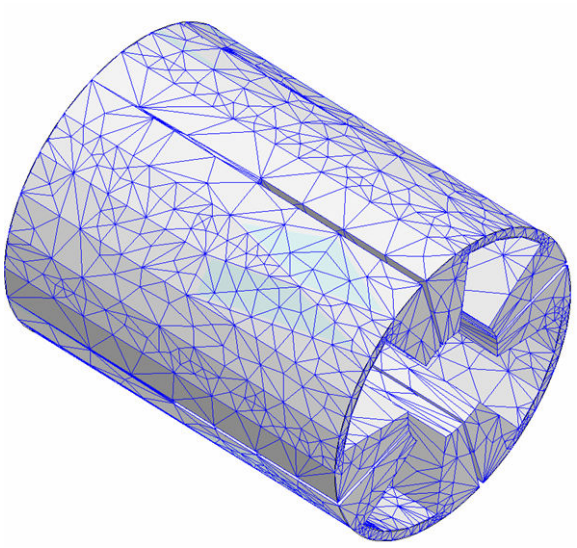
- The geometry is shown above. Dimensions are as follows:
  - Outer radius 160 mm, Ridge diameter 150mm, center diameter 60mm
  - Ridges subtend 45 degrees, evenly spaced at 90 degree intervals
  - Septums subtend 2 degrees and extend half the length of the modeled section (400mm)
  - Two small ‘perturbers’ of ε_r = 1.001 (in red) are inserted at the single-waveguide end to provide stabilization of the degenerate mode order
- The HFSS v9 project was generated by direct translation of the v8.5 project, and solved mesh surface approximation settings.
  - 900 MHz adapt, swept from 400 – 950 MHz
  - 10 passes or to a Delta-S of 0.009
  - 10 and 15 degree normal deviation allowance

Results

- Solution results are shown below, focusing on return loss.
  - Results for 15 degree normal deviation are still unstable
    - Delta-S of 0.0342; final mesh of 16,177 tetrahedra
  - Results for 10 degree normal deviation are excellent
    - Delta-S of 0.00752; final mesh of 15,897 tetrahedra (shown)







Conclusions:

- Due to very narrow annular gaps at the ridge peaks, this geometry type is poorly meshed without user-guidance in the form of tighter surface approximation settings, providing an example for which such guidance is required
- However, with 10 degree faceting the solution convergence was better and to a smaller final mesh than with 15 degree faceting. Therefore in geometric cases where surface approximations are required for a good answer, they do not necessarily imply a longer solution time.

- HFSS v9, 15 degree normal deviation: 10 passes converged to 0.0342 delta-S with a final mesh of 16,177 tetrahedra
  - Solution results similar to prior version, but not there yet; fast sweep indicates further convergence might result in more stable output
- HFSS v9, 10 degree normal deviation: 9 passes converged to 0.00752 with a final mesh of 15,897 tetrahedra
  - Solution results identical to prior version and very clean
  - Shows that surface approximation advice may often parallel that for seeding: a more stringent starting condition and mesh may result in a well-converged solution with a **smaller** ending mesh